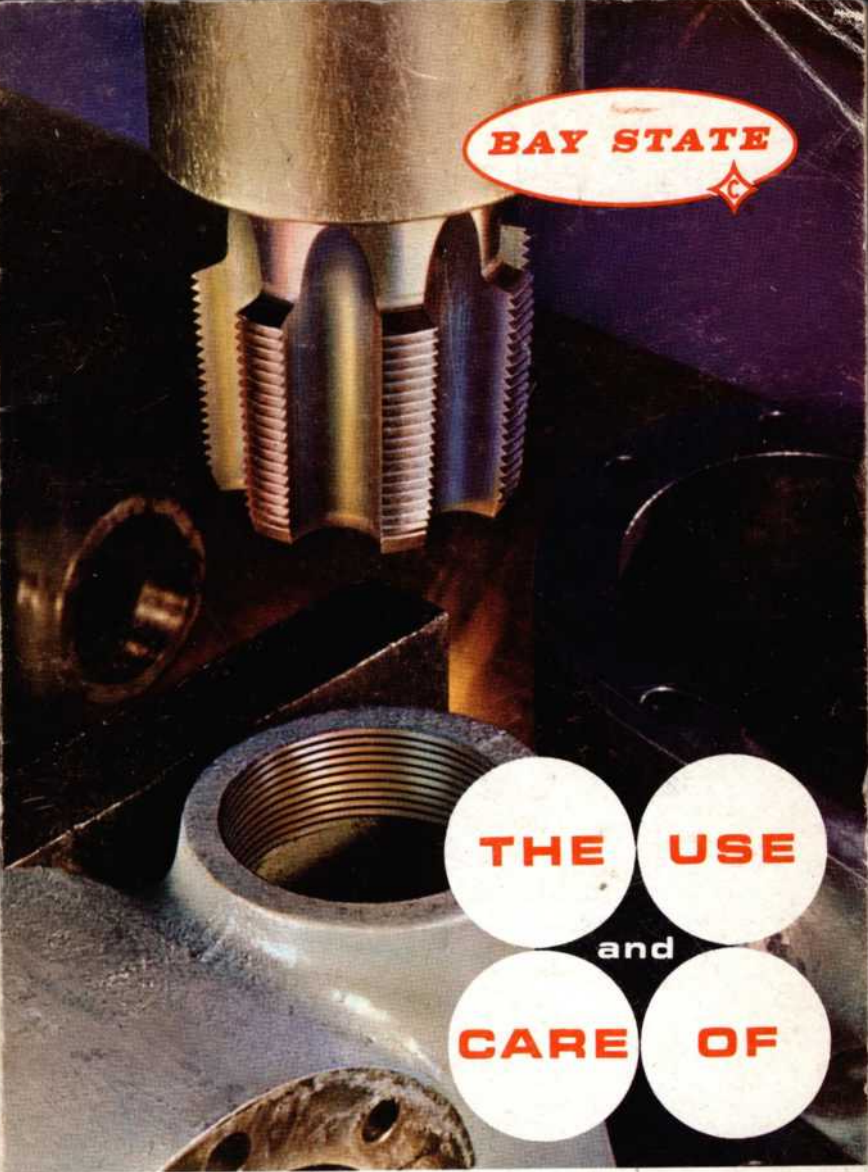




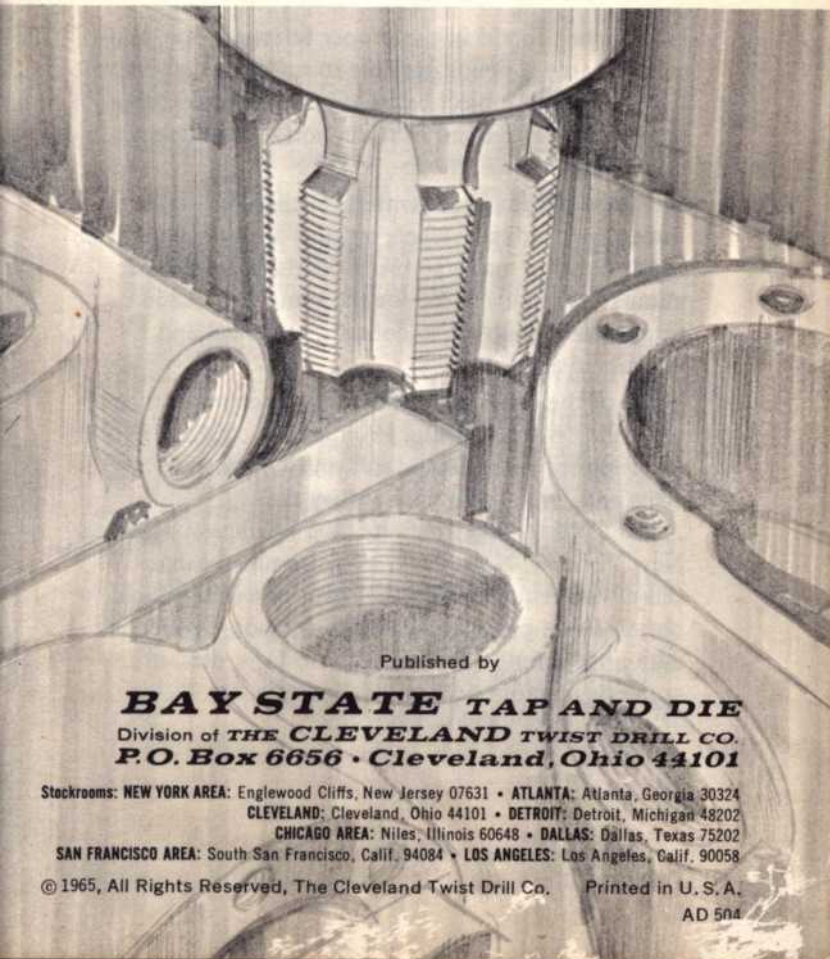
BAY STATE

THE USE
and
CARE OF

TAPS



THE USE and CARE OF TAPS



Published by

BAY STATE TAP AND DIE

Division of **THE CLEVELAND TWIST DRILL CO.**

P.O. Box 6656 • Cleveland, Ohio 44101

Stockrooms: **NEW YORK AREA:** Englewood Cliffs, New Jersey 07631 • **ATLANTA:** Atlanta, Georgia 30324

CLEVELAND: Cleveland, Ohio 44101 • **DETROIT:** Detroit, Michigan 48202

CHICAGO AREA: Niles, Illinois 60648 • **DALLAS:** Dallas, Texas 75202

SAN FRANCISCO AREA: South San Francisco, Calif. 94084 • **LOS ANGELES:** Los Angeles, Calif. 90058

© 1965, All Rights Reserved, The Cleveland Twist Drill Co.

Printed in U. S. A.

AD 504

The Use and Care of Taps

is published to aid in the proper selection, use, and care of taps. It is not possible to make this a complete treatise on the subject because of the complexity of tapping, but rather an introduction to the theories underlying the use of taps.

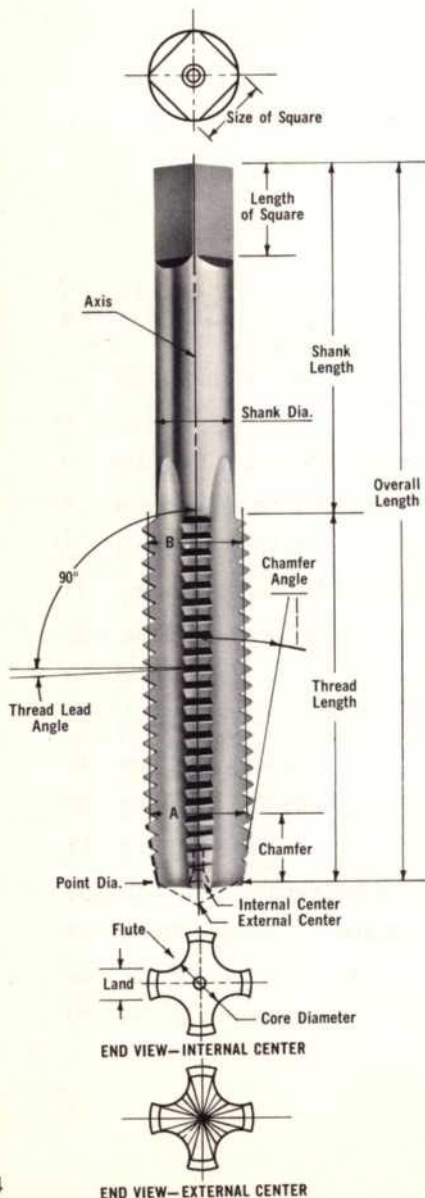
This booklet has been written for distribution throughout industry wherever taps are used. It will be helpful also in manufacturing plants as a reference manual and is available for the same purpose in reasonable quantities to colleges, technical and vocational schools.

Some of the material in this publication was taken from the "Metal Cutting Handbook", "Drilled Holes for Tapping" and "Standards and Dimensions for Taps and Dies," all published by the Metal Cutting Tool Institute, 405 Lexington Avenue, New York 17, New York. We recommend that those who require additional data on cutting tools obtain these publications. They include engineering data and technical information on taps, dies, drills, reamers, counterbores, milling cutters, etc.

Contents

Graphic Terminology	4
Selecting the Tap	6
Classes of Thread and Tap Size	10
Thread Forming of Internal Threads	18
Methods of Tapping (Hand and Machine)	22
Preparation of Holes for Tapping	27
Tapping Speeds	35
Lubrication	38
Surface Treatments	39
Tapping Troubles	40
Resharpening and Grinding	47
Pipe Tapping	55
Tapping Difficult-to-Machine Materials	60
Strength of Tapped Holes	65
Handling and Storage	68
Miscellaneous Tables	70

GRAPHIC TERMINOLOGY



NOTES:

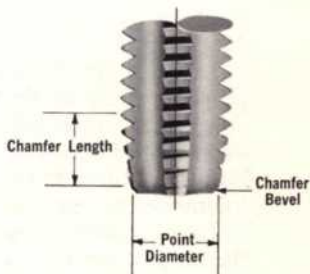
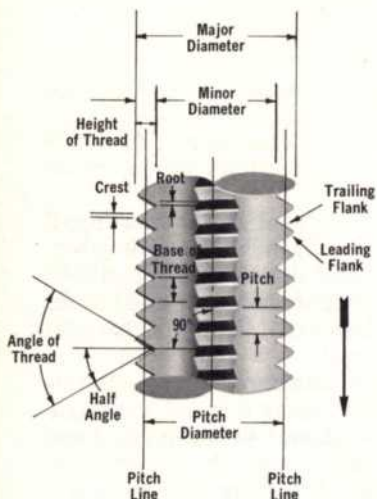
"A"—Pitch diameter at first full thread. This is the correct point for measuring pitch diameter.

BACK TAPER—The amount pitch diameter at "A" is greater than pitch diameter at "B".

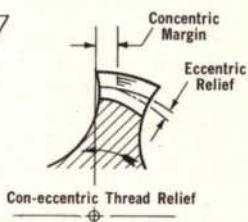
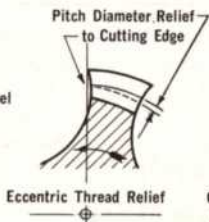
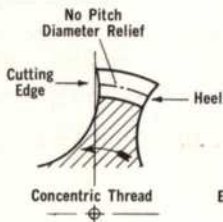


END VIEW—THREE FLUTED
SPIRAL POINTED TAP

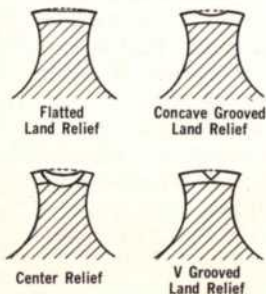
GRAPHIC TERMINOLOGY



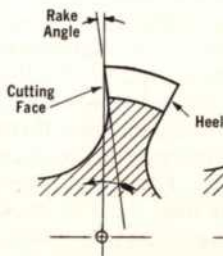
TYPES OF THREAD RELIEF



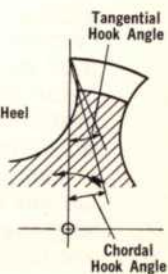
TYPES OF LAND RELIEF



RAKE



HOOK



TAPPING is one of the most difficult machining operations because of the problems of removing chips and providing adequate lubricant at the cutting edge. Furthermore, the relationship between speed and feed is fixed by the lead of the tap and cannot be varied independently.

Tapping problems may be minimized by selecting, if possible, material that is not difficult to tap. Unreasonable lengths of engagement and thread depth should be avoided and adequate clearance should be allowed at the bottom of blind holes. It is important that pitch diameter tolerances be reasonably liberal.

Tapping difficulties may be further avoided by selecting the correct tap for the job and using it in the most suitable equipment. Lubrication should be adequate, speed should be reasonable and all factors should be maintained at peak efficiency. Except in completely automatic cycles, the skill of the operator is of prime importance.

SELECTING THE TAP

Size and style are the two major considerations in selecting the correct tap to use. Size selection is complicated by many variables and is so related to the size of the product thread that it is covered in detail in the section "Classes of Thread and Tap Size" on pages 10-12. Information on style selection appears below.

Selecting the Style of Tap

The first consideration in selecting the optimum style and design is the use for which the tap is intended. The choice will vary depending on whether the tap is to be used for general work with the implied requirement that it must cut many materials reasonably well or whether it will be used on a high production, repetitive operation.

Cut thread taps with the irregularities resulting from heat treatment after threading may be entirely satisfactory if the size limits of the threaded product are not too exacting. For hand tapping operations, taps made of carbon steel may be practical. For the majority of tapping operations, however, high speed ground thread taps are necessary to meet gage specifications.



If the tap is for a production job it preferably should be chosen from the many styles, including special purpose taps, which are considered regular and generally carried in stock. If this is not possible a tap should be used that is specially designed for the job.

The design of special taps involves variation of such features as hook or rake in the flute of the tap, thread relief, length of chamfer and chamfer relief. An important factor in tap performance is the application of the proper surface treatment (see page 38). Whenever special taps are indicated, it is advisable to consult a product application engineer.

Tap troubles are discussed more thoroughly in another section starting on page 40, but it is obvious that some of the common causes of tap failure are directly related to the style of tap which is used. For example; breakage in the tapping of shallow threads due to misalignment may be considerably reduced or eliminated by using a rugged, short flute spiral pointed tap which has short angular flutes at the point and no axial flutes. Chip clogging is one of the most common causes of tap failure. As a means of reducing this potential difficulty, taps are available with spiral points which push the chips ahead, with spiral flutes which draw the chips from the hole and with varying numbers of straight flutes. Generally speaking, the fewer the number of straight flutes in a tap, the greater the chip capacity of each individual flute and the higher the possible hook angle of the cutting face.

The material and physical characteristics of the hole to be tapped have a definite bearing on the style of tap to be selected.

The table on Page 8, showing the many tap styles available, may be used as a guide for selecting the optimum tap for the job.

To obtain the lowest tapping cost, it is frequently necessary to use taps which are designed specifically for the job. If the material to be tapped is soft and gummy, it may be advisable to use taps having a high degree of hook, radial relief in the threads, interrupted threads or a combination of these features. An excessive amount of back taper may

Tap Design Selection Table

MATERIAL	HOLE TYPE	SIZE			
		STRAIGHT THREAD			TAPER PIPE TAPS
		#0 M. S. to #10 M. S.	¼" to ½"	⅝" to 1"	ALL SIZES
Cast Aluminum Forged Aluminum Copper Zinc Die Casting Aluminum Die Casting Magnesium	through	spiral pointed	spiral pointed	4 flute	high speed ground thread for aluminum or steel
	blind	hy-spiral or 2 or 3 flute	hy-spiral or 3 flute	4 flute	
	deep blind	hy-spiral	hy-spiral	4 flute	
Bakelite Cast Brass Bronze Hard Fiber Cast Iron Malleable Iron Plastic Hard Rubber	through or blind	2 or 3 flute bottoming or plug	3 or 4 flute bottoming or plug	4 flute bottoming or plug	high speed ground thread or high speed cut thread
Monel Metal Nickel Silver Cast Steel Machinery Steel Stainless Steel	through	spiral point	spiral point	4 flute	high speed ground thread for steel or high speed ground thread interrupted
	blind	hy-spiral or 2 or 3 flute	hy-spiral or 3 or 4 flute	4 flute	
	deep blind	hy-spiral	hy-spiral	4 flute	
Titanium	SPECIAL TAPS				
Super Strength Alloys High Temperature Alloys	SPECIAL TAPS				

NOTES: Spiral pointed taps may be used for blind holes as well as through holes if there is ample clearance at the bottom for chips.

Two, three and four fluted plug taps are preferable to bottoming taps if full threads are not specified nearer the bottom of the hole than can be tapped with a 3-5 thread plug chamfer.

For Bakelite and Hard Fiber Plastic and Rubber, special multi-flute taps of surface treated high speed steel or special high-speed steel are recommended.

For Deep Blind Holes in any stringy material, FORM taps are very strongly recommended.

Figure 1

The More Common Types of High Speed Ground Thread Taps



SPIRAL POINTED PLUG



SPIRAL POINTED BOTTOMING



4 FLUTE TAPER



4 FLUTE PLUG



4 FLUTE BOTTOMING



**GROUND THREAD
TAPER PIPE**



**GROUND
INTERRUPTED
THREAD
TAPER PIPE**



**GROUND THREAD
TAPER PIPE FOR
STEEL OR ALUMINUM**



2 FLUTE PLUG



2 FLUTE BOTTOMING



3 FLUTE PLUG



3 FLUTE BOTTOMING



HIGH SPIRAL PLUG



HIGH SPIRAL BOTTOMING



FORM TAP

also be helpful but this reduces the number of sharpenings possible before the tap is ground back to the point where the pitch diameter is too small. Variations in the geometry of chamfers and spiral points may also be indicated.

When designing taps for hard material, it is important to minimize the chip load per tooth. This may be accomplished by long chamfers, a greater number of lands, one or more roughing taps or a combination of all three methods. Specifications of the threaded product may not allow long chamfers and this restriction is frequently the factor which necessitates roughing taps.

When helical fluted taps are indicated, the degree of the flute helix may be varied for the best results.

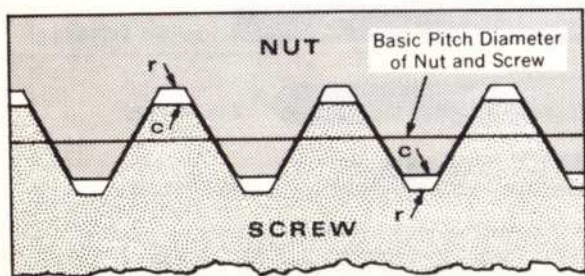
The table, on page 53, shows the approximate amount of chordal hook at the cutting face for tapping some of the materials frequently used.

CLASSES OF THREAD AND TAP SIZE

There is obviously a direct relationship between the size of a tap and the size of the thread which it cuts. "Size" in this section refers to pitch diameter* only.

In an assembly of two threaded parts the looseness or tightness of the fit is determined only by the contact at the flanks of the threads as shown in Figure 2. This contact is controlled by the pitch diameter of both parts.

*Pitch diameter—The diameter of the imaginary co-axial cylinder, the surface of which would pass through the thread profiles at such points as to make the width of the groove equal to $\frac{1}{2}$ of the basic pitch.



Contact is only at flanks. Crests (c) and roots (r) do not touch.

Figure 2

Classes of Thread

When threaded parts are assembled, the internal thread must be large enough to allow the external thread to enter it for the required length of engagement. A second requirement is that the two parts will assemble with a degree of tightness dictated by the use of the fastener.

Before World War I, these goals were accomplished by hand fitting, and an external or internal thread was usually cut to the size of the mating part with which it was to be assembled. Currently, a system of different classes of thread, each of which represents a comparative degree of tightness, has been established and adopted. It provides, for the manufacturers and users of threaded products, a common language of specification and universal interchangeability of internal and external threads of the same class.

These classes of thread designate minima and maxima pitch diameters of internal and external threads. For internal threads, American National Class 2 and Class 3 Threads and Unified Class 1B, 2B and 3B Threads define these limits. The counterparts for external threads are Classes 2, 3, 1A, 2A and 3A. The maximum and minimum pitch diameter of each of these classes for $\frac{1}{4}$ "-20 NC and UNC internal and external threads is shown in Figure 3, page 12.

Classes 1A and 1B threads are intended for use where quick and easy assembly is necessary. Classes 2, 2A and 2B threads are the most commonly used classes of threads for general applications including bolts, screws, nuts and similar threaded fasteners. Classes 3, 3A and 3B are used for applications where closeness of fit and accuracy of lead and angle are important.

Classes of thread actually establish manufacturing tolerances and, since it is generally true that the closer the tolerance the higher the cost involved in producing it, engineers ordinarily select the class of thread with the widest possible tolerance.

Tap Size

Taps rarely cut their own size but produce a thread size (usually oversize) which varies with the material tapped and other conditions (see page 46).

To tap a specified class of thread, several different ground thread tap limits are available. These limits represent small defined variations in tap size. They are designated by limit numbers which have no direct relationship to the above classes of thread. It is unfortunate

Classes of Thread

(Pitch Diameter Limits for $\frac{1}{4}$ -20 NC and UNC)

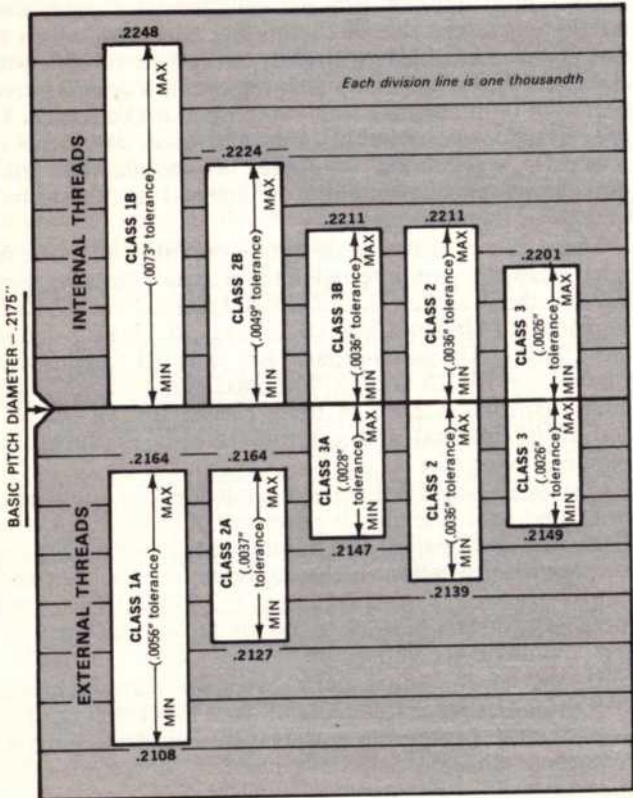


Figure 3

and a source of some confusion that the numerals 1, 2 and 3 are used in both systems. When they are used to define tap limits, however, they are always preceded by an "H" or an "L".



Ground thread tap limits are designated by the letter H (high) above basic pitch diameter or L (low) below basic pitch diameter and a number which establishes the tolerance range in relation to basic. For example, in sizes 1" and smaller, an H1 tap has a tolerance range of from basic to .0005" over basic; an H2 tap from .0005" over basic to .001" over basic, etc.

It will be noted that the tolerance range numeral divided by 2 establishes, in thousandths of an inch, the amount that the maximum tap pitch diameter is above basic in the H series and the amount that the minimum tap pitch diameter is under basic in the L series. The H and L limit number system is outlined in detail below.

Several different limit numbers are available for each diameter pitch combination and it is, therefore, possible to select the one most suitable for the requirement of the tapping operation. The tables in Figures 4a and 4b list recommended limit numbers for different classes of thread.

Pitch Diameter Limits . . .

for Taps to 1" diameter inclusive.

- L1 = Basic to Basic minus .0005
- H1 = Basic to Basic plus .0005
- H2 = Basic plus .0005 to Basic plus .0010
- H3 = Basic plus .0010 to Basic plus .0015
- H4 = Basic plus .0015 to Basic plus .0020
- H5 = Basic plus .0020 to Basic plus .0025
- H6 = Basic plus .0025 to Basic plus .0030

for Taps over 1" diameter to 1½" diameter inclusive.

- H4 = Basic plus .0010 to Basic plus .0020

for Taps not shown above or those over 1½" diameter.

Taps that have H or L limit numbers not shown above or those over 1½" diameter (for example H12 or L10), the limit number following the H or L when divided by 2 indicates in thousandths of an inch the amount the maximum tap pitch diameter is over basic in the H series or the amount the minimum tap pitch diameter is under basic in the L series. In the H series taps the tolerance shown in column D on page 75, subtracted from the maximum pitch diameter will give the minimum pitch diameter. In the L series taps the tolerance shown in column D added to the minimum pitch diameter will give the maximum pitch diameter. These taps will be marked with the appropriate H or L limit number.

As shown in the table below under "Threads Per Inch", two thread series (Coarse and Fine) are readily available. NC and UNC threads provide resistance to stripping of the internal thread. NF and UNF threads are intended for use in thin sections to allow sufficient thread length in the threaded part. They are also used in thin walled components.

Tap Recommendations

FOR CLASSES OF THREAD 2, 3, *1B, †2B AND 3B

UNIFIED AND AMERICAN SCREW THREADS

NOM- INAL SIZE	MACHINE SCREW SIZES										
	THREADS PER INCH		RECOMMENDED TAP FOR CLASS OF THREAD				PITCH DIAMETER LIMITS FOR CLASS OF THREAD				
	NC UNC	NF UNF	Class 2	Class 3	Class †2B	Class 3B	Min. All Classes (Basic)	Max. Class 2	Max. Class 3	Max. Class 2B	Max. Class 3B
0	—	80	G H1	G H1	G H2	G H1	.0519	.0536	.0532	.0542	.0536
1	64	—	G H1	G H1	G H2	G H1	.0629	.0648	.0643	.0655	.0648
1	—	72	G H1	G H1	G H2	G H1	.0640	.0658	.0653	.0665	.0659
2	56	—	G H1	G H1	G H2	G H1	.0744	.0764	.0759	.0772	.0765
2	—	64	G H1	G H1	G H2	G H1	.0759	.0778	.0773	.0786	.0779
3	48	—	G H1	G H1	G H2	G H1	.0855	.0877	.0871	.0885	.0877
3	—	56	G H1	G H1	G H2	G H1	.0874	.0894	.0889	.0902	.0895
4	40	—	G H2	G H1	G H2	G H2	.0958	.0982	.0975	.0991	.0982
4	—	48	G H1	G H1	G H2	G H1	.0985	.1007	.1001	.1016	.1008
5	40	—	G H2	G H1	G H2	G H2	.1088	.1112	.1105	.1121	.1113
5	—	44	G H1	G H1	G H2	G H1	.1102	.1125	.1118	.1134	.1126
6	32	—	G H2	G H1	G H3	G H2	.1177	.1204	.1196	.1214	.1204
6	—	40	G H2	G H1	G H2	G H2	.1218	.1242	.1235	.1252	.1243
8	32	—	G H2	G H1	G H3	G H2	.1437	.1464	.1456	.1475	.1465
8	—	36	G H2	G H1	G H2	G H2	.1460	.1485	.1478	.1496	.1487
10	24	—	G H3	G H1	G H3	G H3	.1629	.1662	.1653	.1672	.1661
10	—	32	G H2	G H1	G H3	G H2	.1697	.1724	.1716	.1736	.1726
12	24	—	G H3	G H1	G H3	G H3	.1889	.1922	.1913	.1933	.1922
12	—	28	G H3	G H1	G H3	G H3	.1928	.1959	.1950	.1970	.1959

*1B tapped holes can be produced with CUT thread taps.

†Cut thread taps in sizes 3—12 NC and NF inclusive may be used under normal conditions and in average materials for producing tapped holes to this classification.

The above recommended taps normally produce the Class of Thread indicated in average materials when used with reasonable care. However, if the tap specified does not give a satisfactory gage fit in the work, a choice of some other limit tap will be necessary.

Tap Recommendations

FOR CLASSES OF THREAD 2, 3, †2B AND 3B

UNIFIED AND AMERICAN SCREW THREADS

FRACTIONAL SIZES

NOM- INAL SIZE	THREADS PER INCH		RECOMMENDED TAP FOR CLASS OF THREAD				PITCH DIAMETER LIMITS FOR CLASS OF THREAD				
	NC UNC	NF UNF	Class 2	Class 3	Class †2B	Class 3B	Min. All Classes (Basic)	Max. Class 2	Max. Class 3	Max. Class 2B	Max. Class 3B
¼	20	—	G H3	G H2	G H5	G H3	.2175	.2211	.2201	.2224	.2211
¼	—	28	G H3	G H1	G H4	G H3	.2268	.2299	.2290	.2311	.2300
⅜	18	—	G H3	G H2	G H5	G H3	.2764	.2805	.2794	.2817	.2803
⅜	—	24	G H3	G H1	G H4	G H3	.2854	.2887	.2878	.2902	.2890
½	16	—	G H3	G H2	G H5	G H3	.3344	.3389	.3376	.3401	.3387
½	—	24	G H3	G H1	G H4	G H3	.3479	.3512	.3503	.3528	.3516
⅝	14	—	G H5	G H3	G H5	G H3	.3911	.3960	.3947	.3972	.3957
⅝	—	20	G H3	G H1	G H5	G H3	.4050	.4086	.5076	.4104	.4091
¾	13	—	G H5	G H3	G H5	G H3	.4500	.4552	.4537	.4565	.4548
¾	—	20	G H3	G H1	G H5	G H3	.4675	.4711	.4701	.4731	.4717
7/16	12	—	G H5	G H3	G H5	G H3	.5084	.5140	.5124	.5152	.5135
7/16	—	18	G H3	G H2	G H5	G H3	.5264	.5305	.5294	.5323	.5308
1	11	—	G H5	G H3	G H5	G H3	.5660	.5719	.5702	.5732	.5714
1	—	18	G H3	G H2	G H5	G H3	.5889	.5930	.5919	.5949	.5934
1 1/8	10	—	G H5	G H3	G H5	G H5	.6850	.6914	.6895	.6927	.6907
1 1/8	—	16	G H3	G H2	G H5	G H3	.7094	.7139	.7126	.7159	.7143
1 1/4	9	—	G H6	G H4	G H6	G H4	.8028	.8098	.8077	.8110	.8089
1 1/4	—	14	G H4	G H2	G H6	G H4	.8286	.8335	.8322	.8356	.8339
1 1/2	8	—	G H6	G H4	G H6	G H4	.9188	.9264	.9242	.9276	.9254
1 1/2	—	12	G H4	G H4	G H4	G H4	.9459	.9515	.9499	.9535	.9516
1 3/4	14 NS	—	G H4	G H2	G H4	G H4	.9536	.9585	.9572	.9609	.9590
2	7	—	G H4	G H4	G H4	G H4	1.0322	1.0407	1.0381	1.0416	1.0392
2	—	12	G H4	G H4	G H4	G H4	1.0709	1.0765	1.0749	1.0787	1.0768
2 1/8	7	—	G H4	G H4	G H4	G H4	1.1572	1.1657	1.1631	1.1668	1.1644
2 1/8	—	12	G H4	G H4	G H4	G H4	1.1959	1.2015	1.1999	1.2039	1.2019
2 1/4	6	—	G H4	G H4	G H4	G H4	1.2667	1.2768	1.2738	1.2771	1.2745
2 1/4	—	12	G H4	G H4	G H4	G H4	1.3209	1.3265	1.3249	1.3291	1.3270
2 3/4	6	—	G H4	G H4	G H4	G H4	1.3917	1.4018	1.3988	1.4022	1.3996
2 3/4	—	12	G H4	G H4	G H4	G H4	1.4459	1.4515	1.4499	1.4542	1.4522

†Cut thread taps may be used under normal conditions and in average materials for producing tapped holes to this classification.

The above recommended taps normally produce the Class of Thread indicated in average materials when used with reasonable care. However, if the tap specified does not give a satisfactory gage fit in the work, a choice of some other limit tap will be necessary.

Figure 4b

The relationship of the different limit numbers to the various classes of thread for $\frac{1}{4}$ "-20 NC and UNC is illustrated in Figure 5. It should be noted that the largest tap shown (H5) could produce the closest class of thread (3) under ideal conditions. It also follows that the smallest limit number (H1) could tap any class of thread if wear life were not an important factor. Thus, the tables in Figures 4a and 4b recommend, for each class of thread, the largest tap which it is possible to use with a reasonable factor of safety.

Comparison of Pitch Diameter Limits

(Classes of Thread and Tap Limits for $\frac{1}{4}$ "-20 NC and UNC)

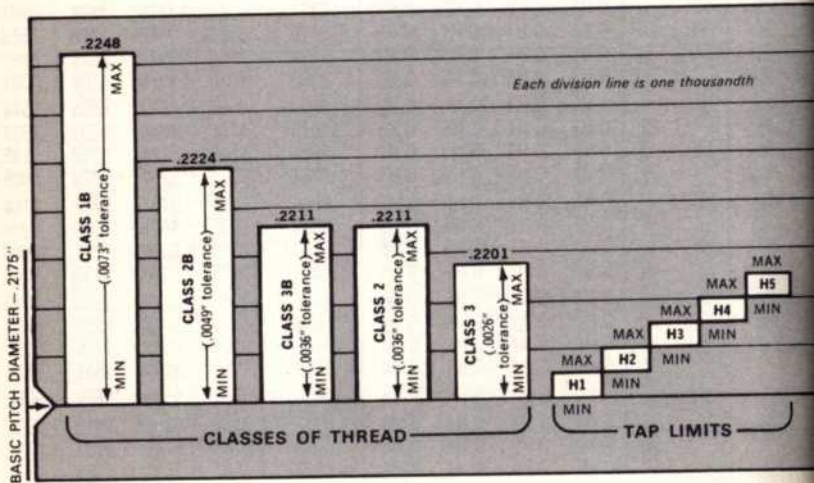


Figure 5

After the limit number has been chosen in accordance with the recommendation, the next step is to prove or disprove this theoretical selection in actual practice. This is necessary because adverse or special conditions may require the use of a higher or lower limit number; for example, a variation in material. Oversize threads are frequently produced when tapping very soft materials; whereas materials which cling to the tap often produce threads which are actually smaller than the tap. This is

because the material expands without being cut properly and contracts after the tap is withdrawn. A thin wall surrounding the thread may effect the same result. Another reason is that improper preparation of holes may tend to produce oversize threads after tapping and burrs in the thread may prevent the "Go" gage from entering, thus falsely indicating an undersize thread. The size of tapped holes is also appreciably affected by the kind of lubricant and the method of applying it.

Probably the most important single factor affecting the size of the internal screw thread is the method of tapping. One or more of the following conditions may result in oversize threads: looseness of spindle, misalignment,



Figure 6
A typical application
showing a High-Spiral
Tap at work in cast steel.

improper chucking of the tap or holding of the part. When tapping by hand or in a machine without a lead screw, the pressure with which the tap is forced into the work has a tendency to shave the flanks of the thread, particularly near the point of entrance. When the tap stabilizes itself after several threads have been tapped, a tapered or bell mouth thread is produced. Occasionally the oversize condition prevails throughout the entire length of thread. Lead screw control to a large extent

eliminates this difficulty. If a lead screw is used, the lead of screw and tap must coincide within reasonable tolerances. If they do not, oversize threads will result.

In conclusion, preliminary size selection can be made from Tap-Limit Class-of-Thread Recommendation Tables, but this selection must be confirmed through actual trial on-the-job.

THREAD FORMING OF INTERNAL THREADS

Internal threads may be generated by threaded, fluteless taps known by various tradenames such as FORM Taps. These taps are held and driven exactly as are conventional taps but, because they have no flutes and do not cut the threads, no chips are produced. Therefore the problems of chip congestion and chip removal often associated with the tapping of blind holes, are eliminated. Because of this, there are many applications of these taps for deep, blind hole tapping in non-ferrous materials and soft steel. A specific use is for tapping parts with wall thickness insufficient to allow cutting of the threads.

In this method of threading, metal actually flows and the minor diameter of the finished thread is smaller than that of the hole before threading. Therefore, to produce a given percentage of thread, the tap drill sizes for these taps must be larger than the tap drill sizes for conventional cutting taps of the same diameter-pitch combinations. Furthermore, the holes to be tapped must be controlled more closely as a hole which is too large will result in a poor thread form whereas a hole which is too small will result in tap breakage. For this reason, reaming may be necessary. A FORM Tap Drill Selector Chart is shown on pages 20 and 21.

Recommended Pitch Diameter Limits

The pitch diameter limits recommended to produce different classes of thread, when using FORM Taps, vary somewhat from those recommended for conventional taps. The table showing these recommendations appears on page 19.

Form Tap Recommendations for Classes of Thread 2, 2B and 3B

TAP SIZE	LIMIT RECOMMENDED		
	CLASS 2	CLASS 2B	CLASS 3B
0-80 NF, UNF	B-2		B-2
1-64 NC, UNC	B-2		B-2
1-72 NF, UNF	B-2		B-2
2-56 NC, UNC	B-2		B-2
2-64 NF, UNF	B-2		B-2
3-48 NC, UNC	B-2		B-2
3-56 NF, UNF	B-2		B-2
4-40 NC, UNC	P-3	P-5	P-3
4-48 NF, UNF	P-3	P-5	P-3
5-40 NC, UNC	P-3	P-5	P-3
5-44 NF, UNF	P-3	P-5	P-3
6-32 NC, UNC	P-3	P-5	P-3
6-40 NF, UNF	P-3	P-5	P-3
8-32 NC, UNC	P-3	P-5	P-3
8-36 NF, UNF	P-3	P-5	P-3
10-24 NC, UNC	P-4	P-6	P-4
10-32 NF, UNF	P-4	P-6	P-4
12-24 NC, UNC	P-4	P-6	P-4
12-28 NF, UNF	P-4	P-6	P-4
1/4-20 NC, UNC	P-4	P-6	P-4
1/4-28 NF, UNF	P-4	P-6	P-4
5/16-18 NC, UNC	P-5	P-7	P-5
5/16-24 NF, UNF	P-5	P-7	P-5
3/8-16 NC, UNC	P-5	P-7	P-5
3/8-24 NF, UNF	P-5	P-7	P-5
7/16-14 NC, UNC	P-5	P-8	P-5
7/16-20 NF, UNF	P-5	P-8	P-5
1/2-13 NC, UNC	P-5	P-8	P-5
1/2-20 NF, UNF	P-5	P-8	P-5

P = Plug Style B = Bottoming Style

The above recommended taps will normally produce the class of thread indicated in most materials. However, if the tap specified does not give a satisfactory gage fit in the work, a choice of some other limit tap will be necessary.

FORM Tap Drill Selector Chart

TAP SIZE	75% Thread			70% Thread			65% Thread		
	Theor. Hole Core Size	Nearest Drill Size	Dec. Equiv.	Theor. Hole Core Size	Nearest Drill Size	Dec. Equiv.	Theor. Hole Core Size	Nearest Drill Size	Dec. Equiv.
0 80	.0536	1.35mm	.0531	.0540	1.35mm	.0531	.0545		
1 64	.0650	1.65mm	.0650	.0655	1.65mm	.0650	.0661		
1 72	.0659	1.65mm	.0650	.0663			.0669	1.7mm	.0669
2 56	.0769	1.95mm	.0768	.0774	1.95mm	.0768	.0781	$\frac{5}{64}$	.0781
2 64	.0780	$\frac{5}{64}$	.0781	.0785	47	.0785	.0791	2.0mm	.0787
3 48	.0884	2.25mm	.0886	.0890	43	.0890	.0898	43	.0890
3 56	.0899	43	.0890	.0904			.0911	2.3mm	.0906
4 40	.0993	2.5mm	.0984	.1000	39	.0995	.1010	39	.0995
4 48	.1014	38	.1015	.1020	38	.1015	.1028	2.6mm	.1024
5 40	.1123	34	.1110	.1130	33	.1130	.1140	33	.1130
5 44	.1134	33	.1130	.1141	2.9mm	.1142	.1150	2.9mm	.1142
6 32	.1221	3.1mm	.1220	.1230	3.1mm	.1220	.1243		
6 40	.1253	$\frac{1}{8}$	.1250	.1260	3.2mm	.1260	.1270	3.2mm	.1260
8 32	.1481	3.75mm	.1476	.1490			.1503	25	.1495
8 36	.1498	25	.1495	.1507	3.8mm	.1496	.1518	24	.1520
10 24	.1688			.1700	18	.1695	.1717	$1\frac{1}{64}$	.1719
10 32	.1741	17	.1730	.1750			.1763		
12 24	.1948	10	.1935	.1960	9	.1960	.1977	5.0mm	.1968
12 28	.1978	5.0mm	.1968	.1989	8	.1990	.2003	8	.1990
$\frac{1}{4}$ 20	.2245	5.7mm	.2244	.2260			.2280	1	.2280
$\frac{1}{4}$ 28	.2318			.2329	5.9mm	.2323	.2343	A	.2340
$\frac{5}{16}$ 18	.2842	7.2mm	.2835	.2861	7.25mm	.2854	.2879	7.3mm	.2874
$\frac{5}{16}$ 24	.2912	7.4mm	.2913	.2927			.2941	M	.2950
$\frac{3}{8}$ 16	.3431	$1\frac{1}{32}$	.3437	.3452	8.75mm	.3445	.3474	S	.3480
$\frac{3}{8}$ 24	.3537	9.0mm	.3543	.3552	9.0mm	.3543	.3566		
$\frac{7}{16}$ 14	.4011			.4035	Y		.4059	$1\frac{1}{32}$	
$\frac{7}{16}$ 20	.4120	Z		.4137	10.5mm		.4154		
$\frac{1}{2}$ 13	.4608			.4634			.4660		
$\frac{1}{2}$ 20	.4745			.4762			.4779		



FORM Tap Drill Selector Chart

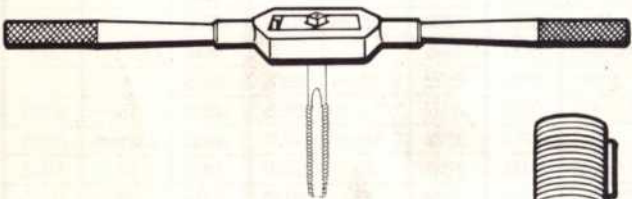
60% Thread			55% Thread			50% Thread			TAP SIZE
Theor. Hole Core Size	Nearest Drill Size	Dec. Equiv.	Theor. Hole Core Size	Nearest Drill Size	Dec. Equiv.	Theor. Hole Core Size	Nearest Drill Size	Dec. Equiv.	
.0549	54	.0550	.0554	54	.0550	.0558	1.4mm	.0551	80 0
.0666			.0672	51	.0670	.0677	51	.0670	64 1
.0673	51	.0670	.0679	51	.067	.0683			72
.0787	47	.0785	.0794	2.0mm	.0787	.0799			56 2
.0796	2.0mm	.0787	.0802			.0807	2.05mm	.0808	64
.0905	2.3mm	.0906	.0913	2.3mm	.0906	.0919			48 3
.0917	2.3mm	.0906	.0924	2.35mm	.0925	.0929	2.35mm	.0925	56
.1018	38	.1015	.1028	2.8mm	.1024	.1035	2.6mm	.1024	40 4
.1035	2.6mm	.1024	.1043	37	.1040	.1049	37	.1040	48
.1148	2.9mm	.1142	.1158	32	.1160	.1165	32	.1160	40 5
.1157			.1166	32	.1160	.1173	32	.1160	44
.1252	1/8	.1250	.1264	3.2mm	.1260	.1274			32 6
.1278	3.25mm	.1280	.1288	30	.1285	.1295	30	.1285	40
.1512	3.8mm	.1496	.1524	24	.1520	.1534	3.9mm	.1535	32 8
.1526	24	.1520	.1537	3.9mm	.1535	.1546	23	.1540	36
.1729	1 1/64	.1719	.1746	17	.1730	.1758			24 10
.1772	16	.1770	.1784	4.5mm	.1772	.1794			32
.1989	8	.1990	.2006	5.1mm	.2008	.2018	7	.2010	24 12
.2014	7	.2010	.2028			.2039	1 3/64	.2031	28
.2295	1	.2280	.2315			.2330	5.9mm	.2323	20 1/4
.2354	1 5/64	.2344	.2368	6.0mm	.2362	.2379	B	.2380	28
.2898	L	.2900	.2917	7.4mm	.2913	.2936			18 5/16
.2955	7.5mm	.2953	.2969	1 9/64	.2969	.2983	7.6mm	.2992	24
.3495	8.9mm	.3504	.3516			.3537	9.0mm	.3543	16 3/8
.3580	T	.3580	.3594	2 3/64	.3594	.3608			24
.4084			.4108			.4132	Z		14 7/16
.4171			.4188			.4205			20
.4686	1 5/32		.4712	12.0mm		.4738	12.0mm		13 1/2
.4796			.4813			.4830	3 1/64		20

METHODS OF TAPPING— HAND AND MACHINE

Taps may be used by hand or in machines. Although fractional and machine screw size taps with squared ends and standardized linear dimensions are traditionally known as hand taps, they are more frequently used in machines than by hand.

Holding Devices

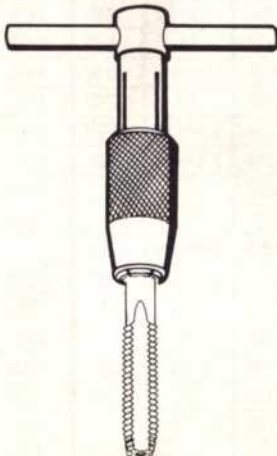
Various methods of holding taps are shown below and on the next page.



Tap Wrench



*Split Sleeve
Tap Driver*



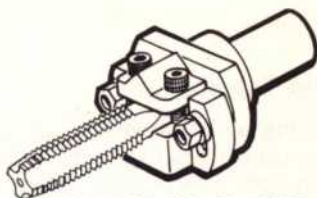
*T-Handle
Tap Wrench*



*Tap Holder with
Horizontal Float*



Special Tap Chuck



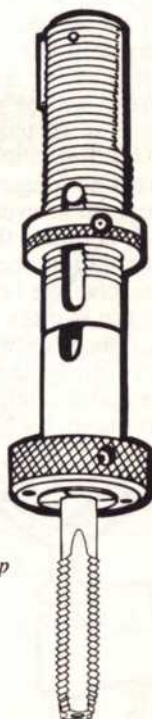
Screw Machine Tap Holder



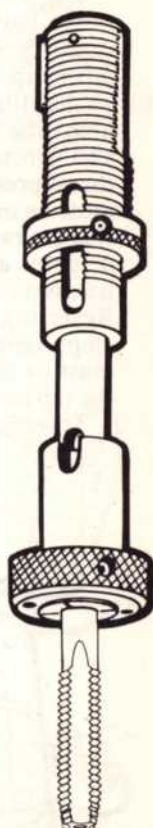
*Tap Holder with
Torque Release*



*Collet Holding Tap
for Assembly
with Holder*



*Tension Type
Tap Holder*



*Compression Type
Tap Holder*

Hand Tapping

Tapping by hand with a tap wrench does not entail the same difficulties experienced when tapping by machine but it does present some different problems. One of these is starting the tap in correct alignment and another is the difficulty, when human reflexes are involved, to transmit a constant flow of power.

When using a tap wrench the square of the tap is fixed in the jaws of the wrench and the chamfered portion is carefully inserted in the hole to be tapped. Usually a taper tap is used as the long chamfer facilitates starting so that the thread will be reasonably concentric with the hole. The thread is started by applying a thrust pressure and a rotary motion simultaneously. Once started, the thrust pressure is no longer necessary but the rotary force must be maintained as evenly and constantly as possible. It is desirable to continue the cut without interruption but if chip accumulation becomes too great, and it is necessary to reverse the tap, it should be done carefully. Reversing the tap is often a cause of failure because the chips remain unbroken when cutting action stops and must be sheared off by the heel of the preceding land as the tap reverses. This often causes chipping and breakage.

Threads produced by conventional hand tapping are

satisfactory for maintenance purposes but, if more accuracy is required, it may be obtained by using a fixture known as a hand taper, such as shown in Figure 7.

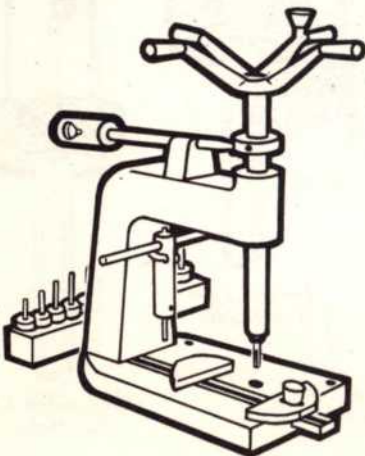


Figure 7



This device acts as a guide and thus insures concentricity. It may utilize lead control which is further insurance against oversize threads.

A common practice that insures better thread quality makes use of any machine tool which provides a center and an adjustable holder to align the tap and hole. The procedure is to chuck the part to be threaded in line with the tailstock center. Next, the internal center at the squared end of the tap (already mounted in a tap wrench) is engaged with the tailstock center and the chamfered end of the tap inserted carefully in the hole. Finally, as a light pressure is continuously kept on the tap by means of the tailstock or feed lever, the tap is started evenly and driven at slow speed. This method is shown in Figure 8.

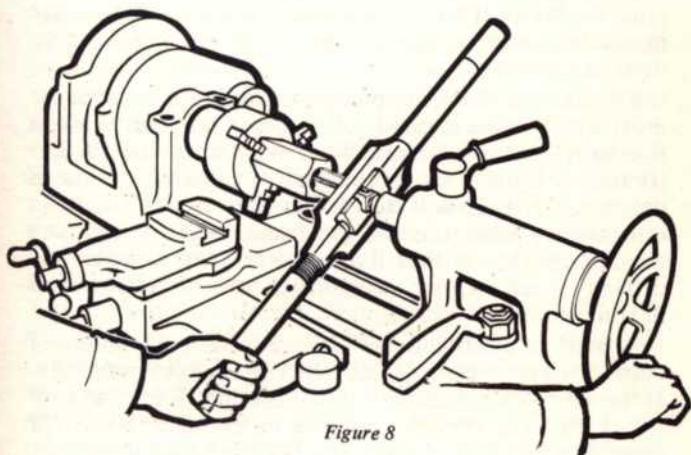


Figure 8

Machine Tapping

Tapping by machine with the correct set-up and fixtures generally produces better threads than those produced by hand tapping. However, the principal reason for machine tapping is to lower tapping costs. This is effected by the higher tapping speeds possible, multiple spindles and automation.

There are problems in machine tapping, not experienced when hand tapping, that are brought about by the much faster speed of the tap. This aggravates the chip disposal problem and develops much more frictional heat which, if excessive, tends to break down the cutting edge of the tap. Correct tap design and proper lubrication are also more important in machine tapping to cut a good thread of correct size.

Unless the threaded portion of the tap can pass completely through the part being tapped, and the work removed from the shank end, the tap must be reversed. This may be done by a friction clutch which drives a right hand tap clockwise when the feed lever is pulled downward toward the work and which reverses the direction of rotation and backs out the tap when the feed lever is pulled upward. If a friction clutch is not used, the change in rotation may be effected by a reversing motor or by reversing gears.

The friction clutch adapted to a drill press is one of the most widely used methods of machine tapping. While it is usually controlled manually, it may be automated. The friction is usually set so that, when excessive torque is developed, the tap will stop rather than break. This is an outstanding advantage of the friction type of tapping chuck. Another is that it usually reverses at twice the driving speed. Other advantages are simplicity and flexibility.

A marked disadvantage, however, is the possibility of cutting oversize threads, particularly in softer materials. If too much or not enough thrust is applied, the flanks of the thread are shaved, resulting in oversize threads or threads which are oversize for only the first few turns. These are known as bell-mouth threads.

The demand for product threads to close tolerances has accelerated the development of positive lead tapping machines with lead screws or cams to control the advance of the tap. When this equipment is used, shaving of the flanks of the product thread is theoretically impossible. However, to offset cam wear, compression or tension holders are frequently used.



PREPARATIONS OF HOLES FOR TAPPING

Holes to be tapped may be either straight or tapered. They may be cored, drilled, bored, punched or extruded with or without reaming. Tapping problems may arise from the physical characteristics of the hole and the condition of the surface to be tapped.

In cored holes, due to certain casting practice, hard spots may be present which may ruin a tap when the first thread is cut. Furthermore, some cored holes may be very abrasive due to scale, sand or other foreign matter, while others may be hard due to chilling. All of these hazards may be aggravated by the draft angles which increase the chip load at the small end of the hole.

Surfaces in drilled or reamed holes occasionally are work hardened, because dull tools were used.

If the hole to be tapped is out of round, the thread will be correspondingly out of round. If the hole is multi-sided the threads will follow the same pattern and this irregularity, known as a wavy thread, affects the true size of the thread. Out of round and wavy taper pipe threads are a common cause of leakage.

When tapping straight threads, the size of the hole is of the greatest importance. For many years a thread depth of 75% has been the rule and many charts have been published showing tap drill sizes which will yield approximately 75% of thread. However, in the new and tougher materials of high tensile strength, it is obvious that a greatly reduced percentage of thread will give more than adequate strength to the fastening.

Figure 9 graphically illustrates the effect of hole size on tapping torque. It should be noted that, in testing the $\frac{3}{8}$ "-16 NC high speed ground thread plug hand tap in AISI 1020 steel, the torque required to drive the tap doubled when the thread percentage increased from 60% to 73% and tripled when the thread percentage was 80%. Tapping problems also increase proportionately.

Effect of Hole Size on Tapping Torque

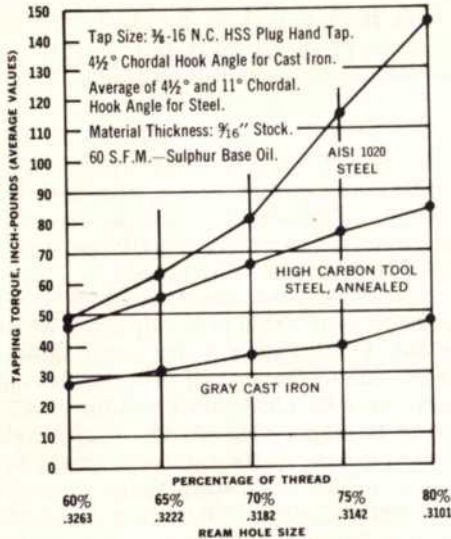


Figure 9

The additional torque required is partially explained by Figure 10 showing the increase in material removed as the per cent of thread is increased. Examination of this chart indicates that, when tapping a 50% thread, 31.3% of the cross section of a full form thread is removed; whereas 60.9%, approximately twice as much, is removed when tapping a 75% thread. The portion of the cross sectional area which does not have to be removed by the tap has been previously removed by the tap drill. A more complete discussion of these two charts appears on pages 16 and 21 of *"Drilled Holes for Tapping"* published by the Metal Cutting Tool Institute (Revised 1959).

Irregularities resulting from hole preparation have a very important influence on tap performance and the quality of the thread which is tapped. Some results of improperly prepared holes, which may be reflected in unsatisfactory threads, are shown in Figures 11 to 19.

Basic 10-24 N.C. Thread

Height .02706 Class 2B Tolerance = .0043

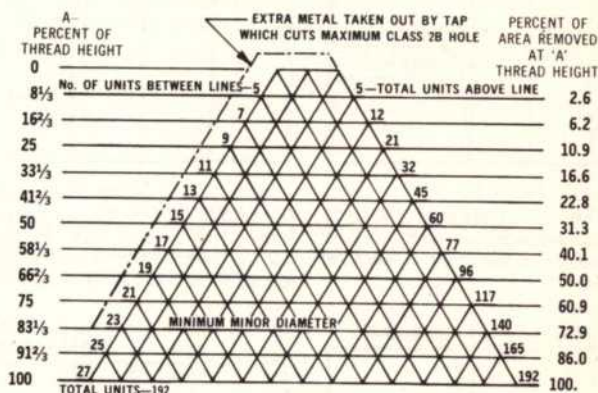


Figure 10

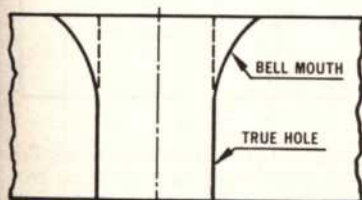


Figure 11

BELL MOUTH OR TAPERED HOLES are holes that are larger in diameter at the orifice than at some distance below. Threads in holes of this type will also be bell mouth.

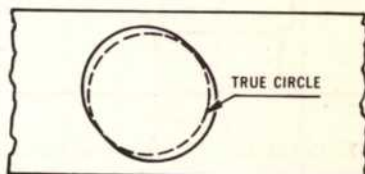


Figure 12

BENT OR CROOKED HOLES are those whose axes are not straight, generally produced by tools "leading off". Threads will tend to follow the hole.

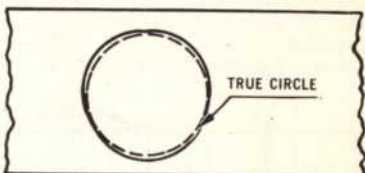


Figure 13

OUT OF ROUND HOLES. Non-circular holes such as these are produced by improperly sharpened drills, or by drills improperly used. Crests of tapped threads will vary in width.

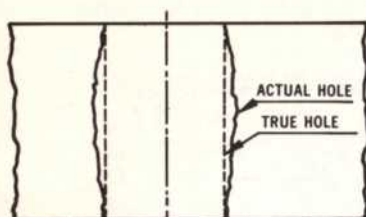


Figure 14

TORN OR "SCORED" HOLES. Holes whose surfaces are badly torn or scored result in a variation in uniform thread size.

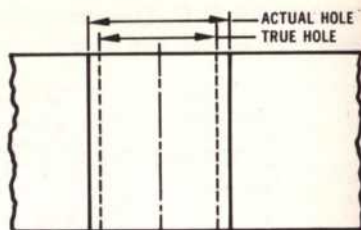


Figure 15

OVERSIZE HOLES are substantially larger in diameter than the tools producing them. When tapped, the percentage of thread will be less than intended.

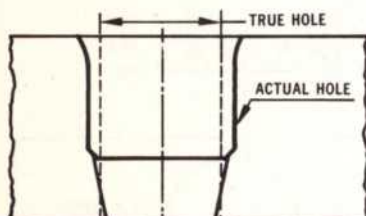


Figure 16

STEPPED HOLES. Holes of two or more diameters produced by a single tool are usually the result of a combination of factors including tools and operating conditions. The effective thread engagement is thus changed.

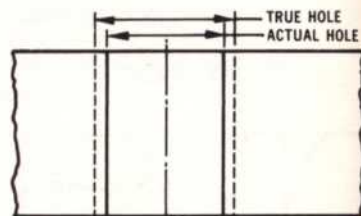


Figure 17

UNDERSIZE HOLES are holes smaller in diameter than the tools which produced them. When tapped, the percentage of thread will be greater than intended.

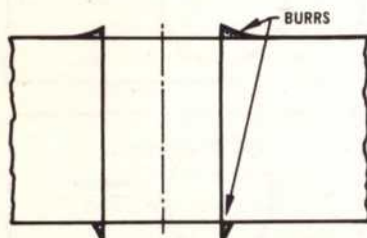


Figure 18

BADLY BURRED HOLES are holes that have an objectionable burr around the edge, either at the top or bottom surface, or both. This is reflected in the quality of the tapped thread.

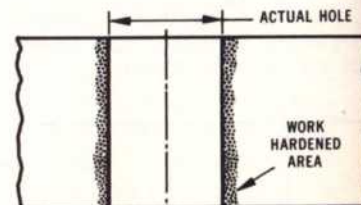


Figure 19

WORK-HARDENED HOLES have a thin surface layer of hard and tough material. This condition is commonly found in holes produced in certain types of work-hardening materials. Dull drills or reamers are usually the cause. It invites tapping troubles.

The smaller the drilled hole, the more serious the chip problem and the greater the amount of torque necessary to drive the tap. It is extremely important to carefully study the relation of drill size to strength of material in order that optimum tap performance may be realized.

The following table shows recommended tap drill sizes for different percentages of thread. Basic dimensions of the internal thread have been used in the calculations. Using the dimensions of the maximum internal thread for different classes of fit would make no appreciable difference in the drill sizes.

Tap Drill Sizes

*Theoretical and Probable Percentage of Full Thread
Produced in Tapped Hole Using Stock Sizes of Drills*

Tap	Tap Drill	Decimal Equiv. of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
0-80	56	.0465	83	.0015	.0480	74
	$\frac{3}{16}$	.0469	81	.0015	.0484	71
1-64	54	.0550	89	.0015	.0565	81
	53	.0595	67	.0015	.0610	59
1-72	53	.0595	75	.0015	.0610	67
	$\frac{1}{16}$	.0625	58	.0015	.0640	50
2-56	51	.0670	82	.0017	.0687	74
	50	.0700	69	.0017	.0717	62
	49	.0730	56	.0017	.0747	49
2-64	50	.0700	79	.0017	.0717	70
	49	.0730	64	.0017	.0747	56
3-48	48	.0760	85	.0019	.0779	78
	$\frac{5}{16}$	.0781	77	.0019	.0800	70
	47	.0785	76	.0019	.0804	69
	46	.0810	67	.0019	.0829	60
	45	.0820	63	.0019	.0839	56
3-56	46	.0810	78	.0019	.0829	69
	45	.0820	73	.0019	.0839	65
	44	.0860	56	.0019	.0879	48
4-40	44	.0860	80	.0020	.0880	74
	43	.0890	71	.0020	.0910	65
	42	.0935	57	.0020	.0955	51
	$\frac{3}{32}$	.0938	56	.0020	.0958	50
4-48	42	.0935	68	.0020	.0955	61
	$\frac{3}{32}$	.0938	68	.0020	.0958	60
	41	.0960	59	.0020	.0980	52
5-40	40	.0980	83	.0023	.1003	76
	39	.0995	79	.0023	.1018	71
	38	.1015	72	.0023	.1038	65
	37	.1040	65	.0023	.1063	58

(Continued on Next Page)

Tap Drill Sizes (continued)

Tap	Tap Drill	Decimal Equiv. of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
5-44	38	.1015	79	.0023	.1038	72
	37	.1040	71	.0023	.1063	63
	36	.1065	63	.0023	.1088	55
6-32	37	.1040	84	.0023	.1063	78
	36	.1065	78	.0026	.1091	71
	$\frac{7}{64}$	.1094	70	.0026	.1120	64
	35	.1100	69	.0026	.1126	63
	34	.1110	67	.0026	.1136	60
6-40	33	.1130	62	.0026	.1156	55
	34	.1110	83	.0026	.1136	75
	33	.1130	77	.0026	.1156	69
	32	.1160	68	.0026	.1186	60
	29	.1360	69	.0029	.1389	62
8-32	28	.1405	58	.0029	.1434	51
	29	.1360	78	.0029	.1389	70
8-36	28	.1405	68	.0029	.1434	57
	$\frac{9}{64}$	.1406	68	.0029	.1435	57
	27	.1440	85	.0032	.1472	79
10-24	26	.1470	79	.0032	.1502	74
	25	.1495	75	.0032	.1527	69
	24	.1520	70	.0032	.1552	64
	23	.1540	67	.0032	.1572	61
	$\frac{5}{32}$	.1563	62	.0032	.1595	56
	22	.1570	61	.0032	.1602	55
	$\frac{5}{32}$	.1563	83	.0032	.1595	75
10-32	22	.1570	81	.0032	.1602	73
	21	.1590	76	.0032	.1622	68
	20	.1610	71	.0032	.1642	64
	19	.1660	59	.0032	.1692	51
	$\frac{11}{64}$	.1719	82	.0035	.1754	75
12-24	17	.1730	79	.0035	.1765	73
	16	.1770	72	.0035	.1805	66
	15	.1800	67	.0035	.1835	60
	14	.1820	63	.0035	.1855	56
	16	.1770	84	.0035	.1805	77
12-28	15	.1800	78	.0035	.1835	70
	14	.1820	73	.0035	.1855	66
	13	.1850	67	.0035	.1885	59
	$\frac{3}{16}$	.1875	61	.0035	.1910	54
	9	.1960	83	.0038	.1998	77
$\frac{1}{4}$ -20	8	.1990	79	.0038	.2028	73
	7	.2010	75	.0038	.2048	70
	$\frac{13}{64}$	.2031	72	.0038	.2069	66
	6	.2040	71	.0038	.2078	65
	5	.2055	69	.0038	.2093	63
	4	.2090	63	.0038	.2128	57

(Continued on Next Page)

Tap Drill Sizes (continued)

Tap	Tap Drill	Decimal Equiv. of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
1/4-28	3	.2130	80	.0038	.2168	72
	7/32	.2188	67	.0038	.2226	59
	2	.2210	63	.0038	.2248	55
5/16-18	F	.2570	77	.0038	.2608	72
	G	.2610	71	.0041	.2651	66
	17/64	.2656	65	.0041	.2697	59
5/16-24	H	.2660	64	.0041	.2701	59
	H	.2660	86	.0041	.2701	78
	I	.2720	75	.0041	.2761	67
3/8-16	J	.2770	66	.0041	.2811	58
	5/16	.3125	77	.0044	.3169	72
	O	.3160	73	.0044	.3204	68
3/8-24	P	.323	64	.0044	.3274	59
	21/64	.3281	87	.0044	.3325	79
	Q	.3320	79	.0044	.3364	71
7/16-14	R	.3390	67	.0044	.3434	58
	T	.3580	86	.0046	.3626	81
	23/64	.3594	84	.0046	.3640	79
7/16-20	U	.3680	75	.0046	.3726	70
	3/8	.3750	67	.0046	.3796	62
	V	.3770	65	.0046	.3816	60
1/2-13	W	.3860	79	.0046	.3906	72
	25/64	.3906	72	.0046	.3952	65
	X	.3970	62	.0046	.4016	55
1/2-20	27/64	.4219	78	.0047	.4266	73
	7/16	.4375	63	.0047	.4422	58
	29/64	.4531	72	.0047	.4578	65
9/16-12	15/32	.4688	87	.0048	.4736	82
	31/64	.4844	72	.0048	.4892	68
	1/2	.500	87	.0048	.5048	80
5/8-11	33/64	.5156	65	.0048	.5204	58
	17/32	.5313	79	.0049	.5362	75
	35/64	.5469	66	.0049	.5518	62
5/8-18	9/16	.5625	87	.0049	.5674	80
	37/64	.5781	65	.0049	.5831	58
	41/64	.6406	84	.0050	.6456	80
3/4-10	21/32	.6563	72	.0050	.6613	68
	11/16	.6875	77	.0050	.6925	71
	49/64	.7656	76	.0052	.7708	72
3/4-16	25/32	.7812	65	.0052	.7864	61
	51/64	.7969	84	.0052	.8021	79
	13/16	.8125	67	.0052	.8177	62
1"-8	55/64	.8594	87	.0059	.8653	83
	7/8	.875	77	.0059	.8809	73
	57/64	.8906	67	.0059	.8965	64
	29/32	.9063	58	.0059	.9122	54

(Continued on Next Page)

Tap Drill Sizes (continued)

Tap	Tap Drill	Decimal Equiv. of Tap Drill	Theoretical % of Thread	Probable Oversize (Mean)	Probable Hole Size	Percentage of Thread
1"-12	$2\frac{9}{32}$	.9063	87	.0060	.9123	81
	$5\frac{9}{64}$	.9219	72	.0060	.9279	67
	$1\frac{5}{16}$	.9375	58	.0060	.9435	52
1"-14	$5\frac{9}{64}$	.9219	84	.0060	.9279	78
	$1\frac{5}{16}$	.9375	67	.0060	.9435	61
	$3\frac{1}{32}$	.9688	84	.0062	.9750	81
1 $\frac{1}{8}$ -7	$6\frac{3}{64}$	.9844	76	.0067	.9911	72
	1"	1.000	67	.0070	1.007	64
	$1\frac{1}{64}$	1.0156	59	.0070	1.0226	55
1 $\frac{1}{8}$ -12	$1\frac{1}{32}$	1.0313	87	.0071	1.0384	80
	$1\frac{3}{64}$	1.0469	72	.0072	1.0541	66
1 $\frac{1}{4}$ -7	$1\frac{3}{32}$	1.0938	84			
	$1\frac{7}{64}$	1.1094	76			
	$1\frac{1}{8}$	1.125	67			
1 $\frac{1}{4}$ -12	$1\frac{5}{32}$	1.1563	87			
	$1\frac{11}{64}$	1.1719	72			
	$1\frac{3}{16}$	1.1875	87			
1 $\frac{3}{8}$ -6	$1\frac{13}{64}$	1.2031	79			
	$1\frac{7}{32}$	1.2188	72			
	$1\frac{15}{64}$	1.2344	65			
1 $\frac{3}{8}$ -12	$1\frac{9}{32}$	1.2813	87			
	$1\frac{19}{64}$	1.2969	72			
	$1\frac{5}{16}$	1.3125	87			
1 $\frac{1}{2}$ -6	$1\frac{21}{64}$	1.3281	79			
	$1\frac{11}{32}$	1.3438	72			
	$1\frac{23}{64}$	1.3594	65			
1 $\frac{1}{2}$ -12	$1\frac{13}{32}$	1.4063	87			
	$1\frac{27}{64}$	1.4219	72			

Reaming
Recommended

TAPPING SPEEDS

The speed at which a tap is operated directly affects the wear life of the tap and the quality of the thread which it produces. Selection of the optimum speed is somewhat restricted because, in tapping, the feed per revolution is fixed by the lead of the thread which eliminates the possibility of varying feed and speed independently as can be done with other cutting tools.

Excessive speed develops extremely high heat which causes early breakdown of the tap at the cutting edge. It also causes loading or galling which is the welding of small particles of the material being tapped to the cutting face or thread flanks of the tap. High cutting speeds restrict adequate lubrication at the cutting zone and aggravate the problem of chip disposal.

As a tap breaks down because of too much speed, the quality of the thread which it produces also deteriorates. Dull taps may produce torn or rough threads, oversize or undersize threads, and objectionable burrs. Taps which are visibly loaded produce rough threads but even a minute amount of load on the flanks, apparent only upon very close inspection, will result in oversize threads.

The speed which will result in the greatest number of holes per tap is not necessarily the optimum speed for the job. It is possible that operating at a higher speed with some sacrifice of tool life, would be more economical because of the lower production time per piece. In other words, a higher tap cost could result in a lower tapping cost.

Determination of the best tapping speed must be influenced by the material to be tapped, physical characteristics of the hole, the fixture, holder, machine and lubricant used in the operation. It must also take into consideration an evaluation of tap cost compared to total threading cost per piece. Therefore, the table of suggested tapping speeds, as shown on page 36, is obviously only a guide to be used as the first step toward the ultimate selection of optimum speed.

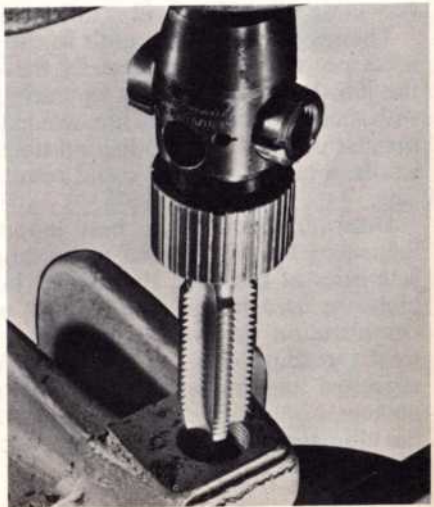
Table 1

Recommended Tapping Speeds

Material	Surface Ft. Per Minute High Speed Taps	Material	Surface Ft. Per Minute High Speed Taps
Aluminum	90—100	Nickel Silver	75—85
Bakelite	60—70	Rubber (Hard)	80—90
Brass	90—100		
Bronze	40—60	Steel, Cast	20—30
Bronze-Manganese	30—45	Chromium	20—30
Copper	90—100	Machinery	40—60
Die Cast, Aluminum	60—70	Manganese	10—15
Zinc	60—70	Molybdenum	20—30
Duralumin	90—100	Nickel	25—35
Fiber	80—90	Stainless	15—25
Iron, Cast	70—80	Tool	25—35
Malleable	35—60	Tungsten	20—30
Monel Metal	20—25	Vanadium	25—35

Table 2 (page 37) is included for a quick conversion of the surface feet shown in Table 1 (above) to the equivalent revolutions per minute for popular tap diameters.

Figure 20
Economically tapping a $\frac{7}{8}$ —14 thread in this motor bracket requires careful selection of tap style and speed.



Speeds for Taps

Conversion Chart—*fpm to rpm*

Table 2

FEET PER MINUTE			10'	15'	20'	25'	30'	35'	40'	50'	55'
Number Size	Fractional Size, Inch	Size Decimal	Revolutions Per Minute								
0		.062	637	955	1230	1540	1850	2150	2460	3080	3390
1		.075	523	785	1020	1275	1530	1780	2040	2550	2700
2		.089	444	666	855	1070	1290	1500	1710	2140	2360
3		.102	386	579	750	940	1125	1310	1500	1880	2060
4		.115	341	512	665	830	995	1160	1330	1660	1825
5	$\frac{1}{8}$	.128	306	459	595	745	895	1045	1190	1490	1640
6		.141	277	415	540	680	815	950	1080	1360	1490
8		.168	233	349	455	570	680	795	910	1140	1250
10		.194	201	302	395	490	590	690	790	980	1080
12		.220	177	265	345	435	520	610	690	870	955
14		.247	158	237	310	385	465	540	620	770	850
	$\frac{1}{4}$	.255	153	229	300	375	450	525	600	750	825
	$\frac{5}{16}$	.318	122	183	240	300	360	420	480	600	660
	$\frac{3}{8}$	.381	102	153	200	250	300	350	400	500	550
	$\frac{7}{16}$	.444	87	131	170	215	260	300	340	430	475
	$\frac{1}{2}$	.507	76	115	150	190	225	265	300	380	415
	$\frac{5}{8}$	.633	61	92	120	150	180	210	240	305	335
	$\frac{3}{4}$	.759	51	76	100	125	155	180	205	255	280
	$\frac{7}{8}$	.885	44	66	85	110	130	155	175	220	240
	1	1.011	38	58	76	95	114	135	155	190	210
FEET PER MINUTE			60'	65'	70'	80'	90'	100'	110'	160'	180'
0		.062	3700	4000	4300	4925	5525	6150	6770	9850	11,050
1		.075	3060	3310	3560	4070	4580	5085	5600	8145	9170
2		.089	2580	2790	3000	3420	3860	4280	4720	6870	7730
3		.102	2250	2435	2620	3000	3360	3760	4120	5980	6230
4		.115	1990	2160	2320	2660	2980	3320	3650	5310	5975
5	$\frac{1}{8}$	.128	1790	1940	2090	2380	2680	2980	3280	4760	5360
6		.141	1630	1760	1900	2160	2440	2720	2980	4320	4880
8		.168	1360	1480	1590	1820	2040	2280	2500	3640	4080
10		.194	1180	1280	1380	1580	1770	1960	2160	3160	3540
12		.220	1040	1130	1220	1380	1560	1740	1910	2760	3120
14		.247	930	1000	1080	1240	1390	1540	1700	2480	2780
	$\frac{1}{4}$	.255	900	975	1050	1200	1350	1500	1650	2400	2700
	$\frac{5}{16}$	.318	720	780	840	960	1080	1200	1320	1920	2160
	$\frac{3}{8}$	.381	600	650	700	800	880	1000	1100	1600	1760
	$\frac{7}{16}$	.444	520	560	600	680	770	860	950	1360	1540
	$\frac{1}{2}$	.507	450	490	530	600	680	760	830	1200	1360
	$\frac{5}{8}$	.633	365	395	425	490	550	610	670	975	1100
	$\frac{3}{4}$	.759	305	330	355	405	460	510	560	815	920
	$\frac{7}{8}$	.885	260	275	305	350	395	435	480	700	785
	1	1.011	230	250	270	305	345	380	420	610	685

LUBRICATION

Adequate lubrication is one of the key elements in a successful tapping operation and therefore must receive more than the casual attention frequently given to it. Cutting fluid serves as a coolant but more importantly as a lubricant.

Certain materials have traditionally been tapped dry but, even where this is possible, improved performance in tap life is generally effected by a judicious selection of some type of cutting oil.

Oxidized surfaces provide a built in lubrication which accounts for the many favorable reports indicating that tap performance has been improved when taps have been surface treated by means of oxidation. The addition of solid lubricants, such as molybdenum disulphide, may improve the effectiveness of the liquid lubricant.

The need and importance of proper lubrication cannot be over emphasized. A table to be used as a general guide for selecting cutting oils for tapping operations is shown below, but specific information and recommendations for each tapping operation should be sought from lubrication engineers.

Suggested Tapping Lubricants

MATERIAL	LUBRICANT	MATERIAL	LUBRICANT
Aluminum	Kerosene & Lard Oil or Kerosene & Light Base Oil	Plastic	
Ampco Metal (Aluminum-Bronze)		Thermoplastic	None or Air Jet
Grades 8-12	Soluble Oil	Thermosetting	None or Air Jet
Grades 16-18	Soluble Oil	Rubber (Hard)	None or Air Jet
Grades 20-24	Soluble Oil	Steels:	
Bakelite	None or Air Jet	Free Machining AISI 1100 Series	Soluble or Sulphur Base Oil
Brass	Soluble Oil or Light Base Oil	Low Carbon (Up to .25%)	Soluble or Sulphur Base Oil
Cast Iron	None or Soluble Oil	Medium Carbon Annealed (.30-.60%)	Sulphur Base Oil
Copper (99% Plus)	Mineral Oil with Lard or Light Base Oil	Heat Treated (.30-.60% Carbon) 225-283 Brinell	Chlorinated Sulphur Base
Fiber	None or Air Jet	Tool—High Carbon & High Speed Steel	Chlorinated Sulphur Base
Magnesium	Light Base Oil Diluted with 50%-40% Kerosene	Stainless	Chlorinated Sulphur Base
Malleable Iron	Soluble Oil or Sulphur Base Oil	Titanium	Chlorinated Sulphur Base
Manganese Bronze	Mineral Oil with Lard or Light Base Oil	Tobin Bronze	Mineral Oil with Lard or Base Oil
Monel Metal	Sulphur Base Oil	Zinc Die Castings	Kerosene & Lard Oil
Naval Brass	Mineral Oil with Lard or Light Base Oil		
Phosphor Bronze	Mineral Oil with Lard or Light Base Oil		

Tapping usually requires a lubricant; the need of a coolant is secondary. The lubricant should be fed to the work under sufficient pressure to insure that it reaches the cutting zone in satisfactory volume as illustrated in Figure 21. If the flow of oil is cycled so that it flows when the tap is cutting and stops when the tap is reversing care should be taken to provide that the lubricant strikes the tap and the work before the tap starts to cut.

Lubrication in most instances requires the lubricant to be clean. Filtering procedures are, therefore, very important and, when the oil is changed, reservoirs, tanks, feed lines, etc. should be cleaned to insure maximum effectiveness of the new oil.

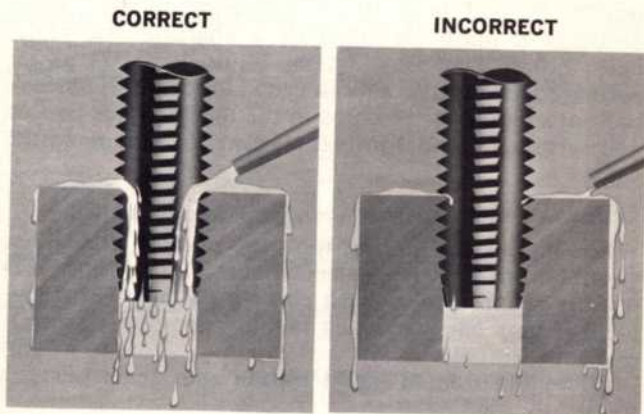


Figure 21

Coolant should be directed to flow, in copious volume, so that it will reach the cutting faces of the tap. Proper flow (left) provides cooling and lubrication and, in most cases, aids in chip removal.

SURFACE TREATMENTS

The efficiency of taps is often improved by the application of one of several different types of surface treatments. When the cause of tap failure or poor threads is loading or galling, the benefits derived from surface treatment may be remarkable. In many instances, surface treatments significantly resist abrasion and generally improve the wear life and performance of all high speed taps.

Three surface treatments—liquid nitride, oxide and chrome plate—are usually available on request from tap manufacturers.

Liquid Nitride

Liquid nitriding produces a very hard, shallow surface on high speed steel tools. This is effected by immersion in cyanide salts at closely controlled temperatures and time cycles. The depth of penetration can be varied to meet specific requirements as, for example, in the tapping of plastics, where a deep case is not likely to cause chipping.

Oxide

Oxide finishes may be developed in air but they are usually applied in a steam tempering furnace which produces a finish identified by a characteristic bluish black color. The oxide retains liquid lubricant at the cutting edges and also acts as a solid lubricant. It tends to prevent welding of the chips to the tool, reduces friction and acts as a thermal insulator between the tap and the work.

Chrome Plate

When high speed cutting tools are chrome plated, the chromium deposit is so shallow that the process is frequently referred to as flash chrome plating. This treatment is very effective when tapping non-ferrous metals and some soft steels.

A combination of liquid nitride and steam oxide is frequently used.

All of these supplemental treatments may be applied to stock tools very quickly.

Surface treatments used judiciously are extremely valuable for extending tap life. To aid in selecting the most suitable applications, a tap manufacturer should be consulted.

TAPPING TROUBLES

Tapping troubles may be divided into two categories: (1) tap failure and (2) imperfections of quality or incorrect size of the product thread. The two troubles are almost always inter-related; for example, the same unfavorable



condition may cause oversize or torn threads in the product, chipping of tap teeth and, finally, tap breakage.

Tap failure may be due to chipping, breakage, loading or excessive wear. Chipping and breakage are the result of identical causes and classified as the same type of failure. Also, chipping frequently leads to a complete fracture of the tap. Loading may lead to breakage. Excessive wear is actually tap failure that results in a poor quality of the product thread produced.

Rough, torn, wavy or chattered threads are the quality imperfections usually encountered in the threaded product. Errors of size are self-definitive.

The following charts illustrate the inter-relationship between tap troubles and product thread troubles. The charts also list the causes as well as the suggested corrective measures.

Tap Troubles

Breaking and Chipping

CAUSE	CORRECTIVE MEASURE
Hitting bottom of hole (or packed chips in bottom)	Use larger drill. Drill hole deeper. Tap part in inverted position. Eject chips by air or hydraulic pressure. Use right hand cut right hand spiral taps to pull chips out of hole. Use FORM Taps.
Chips packing in flutes	Use tap style with more flute space. Be satisfied with shorter engagement or lesser percentage of thread. Select tap style which will eject chips forward or backward.
Misalignment	Align spindle and work. Correct fixture. If only horizontal, use floating holder.
Hard material (Or hard spots)	Anneal. Reduce speed. Lengthen chamfers. Use taps with more flutes.
Improper sharpening	Improve method. Check length of chamfer and degree of relief. Check hook in flutes.

(continued on next page)

Tap Troubles

Breaking and Chipping (Continued)

CAUSE

Binding in hole of thin wall product

Hole too small

Surface of hole work hardened

Hole too shallow

Incorrect lead screw

Machine too heavy

Machine too light

Machine overpowered

Machine underpowered

Feed pressure too great

Power transmission interrupted

Wrong tap

(a) flute space inadequate

(b) Insufficient chamfer

(c) Wrong hook of cutting face

Lubricant inadequate

CORRECTIVE MEASURE

Alter tap to cut more freely if possible. Improve lubrication. Check clamping device to see that hole is not distorted. Reduce land width. Relieve land.

Use larger tap drill.

Check previous operations to insure use of sharp tools. Change material or improve hole preparation.

Check casting or machining operation which may be faulty.

Use correct lead screw.

Correct condition.

Correct condition.

Correct condition.

Correct condition.

Shorten feed lever or instruct operator.

Correct slipping belt or other power transmission trouble.

Substitute tap with more adequate flute space such as 2 or 3 fluted tap instead of 4 fluted tap. Use spiral pointed or spiral fluted tap.

Lengthen chamfer.

Regrind with correct hook.

Use more suitable lubricant. Apply a greater flow with more pressure at the cutting zone.

Tap Troubles

Breaking and Chipping (Continued)

CAUSE

CORRECTIVE MEASURE

Tapping too fast

Reduce speed.

Tap cutting too close to size

Use freer cutting tap; i.e., with more hook, relief or shear.

Tap geometry incorrect.

Change hook or chamfer relief. Use tap with thread relief.

Affinity of product metal for tap material

Apply any corrective measures above. Lubrication and surface treatment most important.

Excessive Wear

Abrasive material

Change material if possible. Improve lubrication. Use liquid nitride or chrome surface treatment.

Abrasive surface of hole due to casting

Improve casting procedure if possible. Improve lubrication. Use nitride or chrome surface treatment.

Improper tap style

Change to more suitable tap style.

Insufficient lubrication

Use more suitable lubricant. Apply more copiously with pressure at cutting zone.

Loading

Correct causes of loading. (See below)

Loading

Insufficient lubrication

Change to more suitable lubricant. Apply more copiously and with pressure at cutting zone.

Excessive speed

Reduce tapping speed.

Wrong style of tap

Use freer cutting tap; i.e., spiral point, spiral flute, interrupted thread, radially relieved. *Surface treatment very important.*

(continued on next page)

THE ILLUSTRATIONS BELOW SHOW SOME OF THE MORE COMMON CAUSES OF TROUBLE

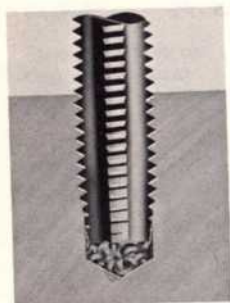


Figure 22

Chips packed at the bottom of hole may cause chipping and breakage. Correct by using shallower thread depth or drilling holes deeper to allow more clearance. A right-hand spiral-fluted tap or a FORM tap might help.

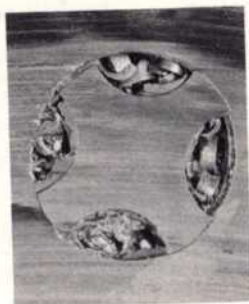


Figure 23

Chips packed in flutes develop excessive tapping torque; may cause oversize threads, chipping or fracture of tap. Correct by using tap with more chip room, larger tap drill, right-hand spiral fluted tap or FORM tap.

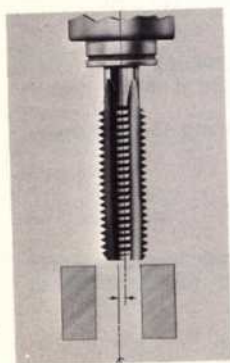


Figure 24

Horizontal misalignment may be eliminated by holder with horizontal float.



Figure 25

Misalignment where axis of tap is not in line with axis of hole to be tapped. Eliminated by adjusting work or fixture.

Product Thread Trouble

Torn or Rough Threads

CAUSE	CORRECTIVE MEASURE
Dull tap	Use sharp tap.
Chip congestion	Use tap with more chip room. Use lesser percentage of thread. Drill hole deeper. Tap shorter thread depth if possible. Use tap which will eject chips.
Incorrect tap design	Regrind tap with correct hook at cutting face (Usually increased). Consider change in thread relief or tap with shearing cut.
Loading	Correct causes of loading. (See page 43).
Inadequate lubrication	Use more suitable lubricant. Apply a greater flow with more pressure at cutting zone.
Hole improperly prepared	Torn areas on the surface of the drilled, bored or cast hole will be reflected in the minor diameter of the thread which is tapped. Improve hole preparation, avoid work hardening.

Wavy Threads

Loose spindle	Repair spindle, clamp work rigidly.
Incorrect chamfer	Correct chamfer grind (usually by less relief)
Incorrect thread relief	Use tap with correct relief (usually less or no radial relief).
Hole out of round	Prepare hole properly.

Undersize Threads

Pitch diameter of tap too small	Use larger pitch diameter (higher H number).
---------------------------------	--

(continued on next page)

Product Thread Trouble

Undersize Threads (Continued)

CAUSE	CORRECTIVE MEASURE
Excessive speed	Reduce speed.
Thin wall material	Use tap which cuts as freely as possible. Usually more hook or shear. Improve lubrication. If thin wall part, hold this portion of work in a completely restricting clamp to prevent it from expanding while being tapped. Use oversize tap.
Dull tap	Sharpen tap.
Burr on thread	Use sharper tap.

Oversize or Tapered (Bell Mouth) Threads

Loose spindle or worn holder	Replace or repair spindle or holder.
Excessive float in spindle or fixture	Reduce float.
Misalignment	Align spindle, fixture and work.
Excessive thrust pressure	Adjust pressure, use lead screw, use care if manual, shorten feed lever.
Loading	Correct causes of loading.
Tap too large (wrong H number)	Use smaller tap.
Dull tap	Sharpen tap.
Chip packing in flutes	Use different style with deeper flutes, spiral flutes or spiral points.
Too much radial relief in threads of tap	Use tap with less or no radial relief.



Removing Broken Taps

Ideal tapping conditions may never be completely realized. Tap breakage will occasionally occur and, when it does, removing that portion of the tap remaining in the work piece becomes a problem.

Taps usually fracture flush with or below the orifice of the hole being tapped. If breakage does not take place there and a substantial length of the broken tap is accessible, removal is a comparatively simple procedure.

When breakage occurs below the orifice, removal is more difficult. If the portion remaining in the hole is not wedged too tightly, it may be removed by a tap extractor. This is a device with prongs which are forced into the flutes, to provide a grip for removal of the broken tap.

However, it is frequently impossible to use a tap extractor because of chip packing, wedging or shattering of the remaining portion of the tap in the hole. Under these conditions, carbon steel taps may be removed by means of a screw extractor after annealing and drilling. It is impracticable to follow this procedure with high speed taps because they cannot be readily annealed. For this reason when the value of the component or product before tapping is relatively high, removal can be accomplished by electrical discharge disintegration.

RESHARPENING AND GRINDING

When to Sharpen Taps

To obtain maximum wear life from taps, they should be sharpened at the very earliest indication of dullness. If tapping is being performed by hand, or by a friction device sensitive to the feel of the operator, the first sign of dullness appears as an increase in the amount of torque necessary to drive the tap. This is often referred to as "pulling hard". If the tapping operation is without manual control, perhaps operated automatically with a lead screw, the first tangible signal of tap breakdown will probably come from some noticeable deterioration in the product thread. It is in this type of repetitive operation that taps should be removed before there is any perceptible breakdown and sharpened in accordance with a predetermined cycle. Then, only a very light

chamfer grind is necessary which insures optimum performance and maximum economy.

How to Sharpen Taps

Taps may be sharpened by regrinding the chamfer and, in cases of extreme wear, it is also necessary to grind the cutting face. Chamfer grinding can be performed by hand or by the use of one of several chamfer grinding attachments which can be mounted on universal tool or cutter grinders or similar equipment.

Taper, plug and bottoming chamfers are shown in Figure 26. Correct and some incorrect chamfers are shown in Figure 27.

TAP CHAMFERS



TAPER
8 to 10 threads



PLUG
3 to 5 threads



BOTTOMING
1½ to 2 threads

Figure 26

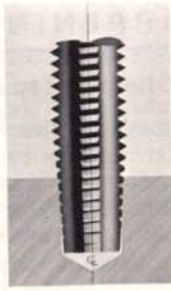
HOW CHAMFER AFFECTS TAPPED-HOLE QUALITY



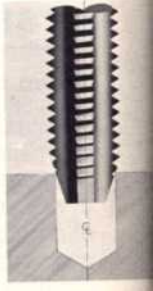
A



B



C



D

(A) Correct chamfer cuts quality threads and provides long tool life, (B) Uneven chamfer produces rough, oversize threads and increases tap breakage, (C) When the chamfer is too long, the tap will hit bottom before the required length of a full thread is cut, and (D) Chamfer ground with a point too small and incorrect angle reduces number of effective teeth, increases chip load.

Figure 27

Chamfer Grinding Taps by Hand

If equipment for chamfer grinding by machine is not available, the alternative is hand sharpening. This method requires no equipment except a grinding head. It is fast but cannot be expected to be as accurate as machine grinding.

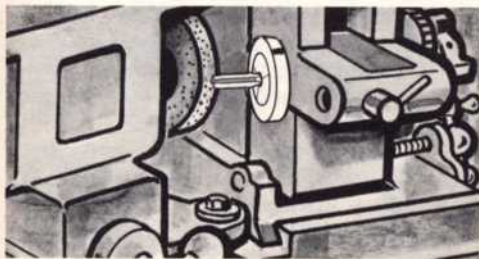
The customary method of sharpening chamfers by hand is shown in Figure 28. The shank of the tap is held firmly, between the thumb and index finger of the left hand and the tap is positioned at the desired angle to the face of the wheel. The thumb and forefinger of the right hand hold the threaded portion against the wheel to steady it. Then, with a cam like motion of the left wrist, the chamfered portion of each land is fed into the wheel, with increasing pressure, from cutting edge to heel. This movement produces the chamfer relief.



Figure 28

Chamfer Grinding by Machine

Chamfer grinders hold the tap at the desired angle in a collet or on centers, locate the high point of the chamfer on one land and then grind relief on the chamfer of every land by a cam action. An example of this attachment is shown in Figure 29. This type of equipment is particularly suitable if the land is accurately indexed. If not, some other method of location, such as a finger, is more satisfactory. Taps may be ground on centers but this obviously is not possible when repeated chamfer grinds have made

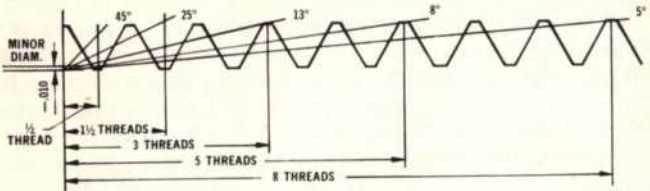


it necessary to grind back the end of the tap, thereby removing the center.

Figure 29

For grinding chamfers, the original chamfer relief should be followed but it can be varied to cope with different materials and job conditions.

Chamfer angles and numbers of threads chamfered for the various conventional chamfers are shown below.



Number of Threads Chamfered on Tap for Different Chamfer Angles.

Chamfer relief for most materials and conditions should be between 4 and 6 degrees. In $\frac{1}{8}$ " of land width, a 4° chamfer amounts to a drop off of .0087"; whereas a 6° chamfer results in a drop off of .0131". Due to the fact that chamfered threads follow a helical path, more relief for tapping the same materials should be ground on bottoming chamfers than on plug chamfers.

When sharpening taps, it should be noted that the diameter at the small end of the chamfer should be at least .005" smaller than the hole to be tapped.

Flute Grinding

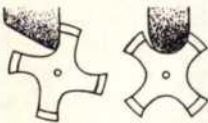


Figure 30

When taps are sharpened along the cutting faces of the lands, a 60 to 80 grit wheel should be formed, as nearly as possible, to the original flute shape. If the flute is then ground with the wheel in the position shown in Figure 30, the

cutting edges of all teeth are restored with the correct angle. This operation can be performed by hand but greater accuracy and uniformity can be realized by machine or fixture grinding. A simple method is to mount the tap between centers on the universal grinder and to pass the tap back and forth under the grinding wheel. The method of grinding the flute vertically is shown in Figure 31.



Figure 31

By the same procedures, new or used taps can be adapted by grinding to different tapping conditions. The following grinding wheels are recommended:

- | | |
|---|--------------------------------------|
| 1. For grinding the chamfer | { 60 Grit
K Hardness
Vitrified |
| 2. For grinding the flutes
of taps over $\frac{1}{2}$ " | { 60 Grit
K Hardness
Vitrified |
| 3. For grinding the flutes
of taps $\frac{1}{2}$ " and smaller | { 60 Grit
P Hardness
Rubber |

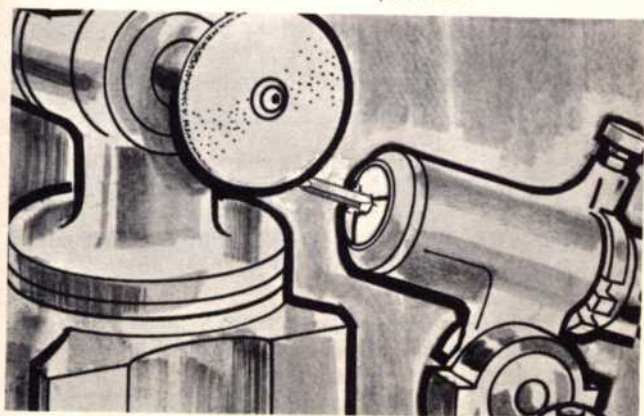


Figure 32

The angular flutes of spiral pointed taps may be ground by hand. However, machines which are available to perform this operation provide greater accuracy. Figure 32 shows a special machine set-up for grinding the angular flutes of spiral pointed taps.

Whether spiral pointed taps are sharpened by machine as shown in Figure 33 or by hand, the following basic procedure should follow:

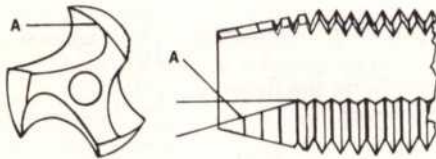


Figure 33

1. Obtain a new spiral pointed tap of the same size and pitch to use as a sample.
2. Select a grinding wheel of similar specifications to that recommended on page 51 for grinding the chamfer.
3. Dress the wheel to a convex shape to conform to the configuration of the flute shape in the sample spiral point.
4. Regrind the spiral point with light grinding passes being careful to duplicate the shear angle "A" on Figure 33.
5. Regrind the chamfer as previously outlined.

Page 53 shows the approximate degree of hook or rake which is suitable for various materials.

Page 54 shows the amount of offset of the flute grinding wheel to achieve the required rake or chordal hook.

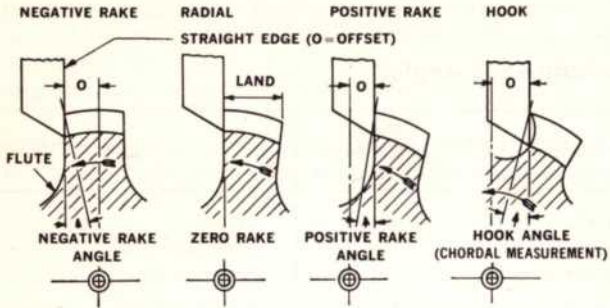
Cutting Face Angles

Material	Rake or Chordal Hook Angle	Material	Rake or Chordal Hook Angle
Aluminum	10° to 20°	Plastic:	
Ampco Metal: (Aluminum-Bronze)		Thermoplastic.	5° to 8°
Grades 8—12	9° to 12°	Thermosetting	0° to 3°
Grades 16—18	5° to 8°	Rubber (Hard)	0° to 3°
Grades 20—24	0° to 3°	Steels:	
Bakelite	0° to 3°	Free Machining	
Brass	2° to 5°	AISI 1100 Series	9° to 12°
Cast Iron	0° to 3°	Low Carbon	
Copper (99% Plus)	15° to 18°	(Up to .25%)	9° to 12°
Fiber	5° to 8°	Medium Carbon	
Magnesium	10° to 20°	Annealed (.30—60%)	7° to 10°
Malleable Iron	5° to 8°	Heat Treated (.30—60% Carbon) 225-283	
Manganese Bronze.	9° to 12°	Brinell	5° to 8°
Monel Metal	9° to 12°	Tool-High Carbon & High Speed Steel	5° to 8°
Naval Brass	5° to 8°	Stainless	10° to 15°
Phosphor Bronze	9° to 12°	Titanium	9° to 12°
		Tobin Bronze	5° to 8°
		Zinc Die Castings	12° to 15°

These recommended cutting face angles may be subject to a variation of 3° to 5° depending on size, pitch and design of flute.

The cutting face angle recommendations do not represent a tolerance. They show the variable range for the material listed. The manufacturing tolerance would be applied to the angle selected.

Offset for Grinding Tap Flutes



SIZE OF TAP	ANGLE OF RAKE OR CHORDAL HOOK															
	1°	2°	3°	4°	5°	6°	7°	8°	9°	10°	12°	14°	16°	18°	20°	
	OFFSET FROM CENTERLINE OF TAP															
#6	.001	.002	.004	.005	.006	.007	.009	.010	.011	.012	.015	.018	.020	.023	.026	
#8	.001	.003	.004	.006	.007	.009	.010	.012	.013	.015	.018	.021	.024	.027	.030	
#10	.002	.003	.005	.007	.008	.010	.012	.014	.015	.017	.020	.024	.028	.031	.035	
#12	.002	.004	.006	.008	.010	.012	.014	.015	.017	.020	.023	.028	.032	.036	.040	
¼	.002	.004	.007	.009	.011	.013	.016	.018	.020	.022	.027	.032	.037	.041	.046	
⅜	.003	.006	.008	.011	.014	.017	.020	.022	.025	.028	.034	.040	.046	.052	.058	
½	.003	.007	.010	.013	.017	.020	.023	.027	.030	.033	.040	.047	.055	.062	.069	
⅝	.004	.008	.012	.016	.019	.023	.027	.031	.035	.039	.047	.055	.064	.070	.081	
¾	.004	.009	.013	.018	.022	.027	.031	.035	.040	.045	.054	.063	.073	.082	.092	
⅞	.005	.010	.015	.020	.025	.030	.035	.040	.045	.050	.061	.071	.082	.093	.104	
1	.006	.011	.017	.022	.028	.033	.039	.044	.050	.056	.067	.079	.091	.103	.115	
1¼	.007	.013	.020	.027	.033	.040	.047	.053	.060	.067	.081	.095	.109	.123	.138	
1½	.008	.015	.023	.031	.039	.046	.054	.062	.070	.078	.094	.110	.127	.144	.161	
1¾	.009	.018	.026	.035	.044	.053	.062	.071	.080	.089	.107	.126	.144	.164	.184	
2	.010	.020	.030	.040	.050	.060	.070	.080	.090	.100	.121	.142	.163	.185	.207	
2¼	.011	.022	.033	.044	.055	.066	.077	.089	.100	.111	.134	.157	.181	.205	.230	
2½	.012	.024	.036	.049	.061	.073	.085	.098	.110	.122	.147	.173	.199	.225	.252	
2¾	.013	.026	.040	.053	.066	.080	.093	.106	.120	.133	.161	.189	.217	.246	.276	
¾ Pipe	.003	.007	.010	.013	.017	.020	.023	.027	.030	.033	.040	.047	.054	.061	.069	
¾ "	.005	.009	.014	.019	.023	.028	.033	.038	.042	.047	.057	.067	.077	.087	.097	
¾ "	.006	.012	.018	.024	.029	.035	.041	.047	.053	.059	.071	.084	.096	.109	.122	
½ "	.007	.015	.022	.029	.037	.044	.051	.059	.066	.074	.089	.104	.120	.136	.152	
¾ "	.009	.018	.027	.037	.046	.055	.064	.074	.083	.092	.111	.130	.150	.170	.190	
1 "	.011	.023	.034	.046	.057	.069	.080	.092	.103	.115	.139	.163	.188	.212	.238	
1¼ "	.015	.029	.043	.058	.072	.087	.101	.116	.131	.146	.176	.206	.237	.269	.302	
1½ "	.017	.033	.050	.066	.083	.100	.116	.133	.150	.167	.202	.236	.272	.308	.345	



PIPE TAPPING

Pipe threads, as the name implies, are for joining units of systems intended to accommodate the flow of liquids or gases, such as water, steam, fuel or compressed air. Although there are some applications where internal straight pipe threads are assembled with external taper pipe threads, the great majority of internal pipe threads are tapered and, because of this, the problems inherent in the tapping of them are much more severe than with straight threads.

The thread form of the American Standard Taper Pipe Thread (NPT) is identical to the thread form of the American National Thread except for the truncation, which is only approximately $1/3$ that of the American National Form. The principal difference between the two threads is the fact that the pipe thread is tapered $3/4''$ per foot. However, a line perpendicular to the base of each thread is also perpendicular to the axis.

When tapping taper pipe threads, every tooth of the tap engaged with the work is cutting until the rotation is stopped. This takes much more torque than does the tapping of a straight thread during which only the chamfered teeth and the first full thread are actually engaged in removing metal. In all tapping, except through hole tapping, reversing the tap is frequently the cause of chipping, poor threads and other mal-functions and thread damage. This is because each tooth, when rotation stops, has raised a chip which must be broken or perhaps partially wedged into the thread by the heel of the preceeding land as the tap is reversed.

In straight threaded taps, this condition exists only at and slightly beyond the chamfered portion; whereas in taper threaded taps, it exists throughout the length of engagement of the tap and the work. The problem is further aggravated in taper pipe tapping by the fact that all threads have eccentric relief at major, pitch and minor diameter. This is necessary because of the increase in diameter from the cutting edge to the heel of each tooth. Eccentric relief over and above this amount is customary to minimize the high torque requirements.

Because of the fact that the heel of each cutting tooth must reverse against a chip, chipping of tap teeth is one of the most common causes of failure encountered in

taper pipe tapping. Furthermore, since all engaged teeth are cutting at the same time, a stress is set up along the cutting edges of the pipe tap. This can cause fracture lines and subsequent chipping when the tap is reversed.

In a taper pipe tapping operation, a stop mark is left by every tooth at the point where rotation stops and, when the thread is examined from the large end, an accumulation of these marks gives the appearance of a line where every land has stopped cutting. Using a spiral fluted tap in no way eliminates these stop marks, although it may appear to do so from visual inspection only. They cannot be avoided and can be removed only by burnishing or some subsequent operation. The lines may be objectionable from a rigid inspection standpoint but they very seldom are the cause of leakage.

Since the cross sectional area of a taper pipe tap increases as the tap advances into the fitting, complete fracture of pipe taps is a rarity. Tooth chipping, for the reasons mentioned above, is usually the cause of final failure.

The ultimate test of taper pipe threads is whether or not they will stand the pressures to which they must be subjected without leaking. All of the tapping problems inherent in tapping straight threads are present in the tapping of taper pipe threads. However, they are greatly aggravated not only because the thread is tapered but by rigid thread specifications which are established to insure pressure tight joints. All of these factors necessitate extreme care and attention to all details, such as selection of properly designed taps, speed, lubrication and equipment.

To lessen the damaging effects of chip wedging on both forward and reverse rotations, the interrupted thread pipe tap was developed. This tap, shown in Figure 34, (compared to the full thread tap shown in Figure 35) has every other tooth removed on each land with alternate teeth removed on adjacent lands so that the pattern, starting at any point in the thread and following around the helix, is tooth, space, tooth, space, tooth, space, etc.



Figure 34

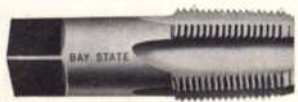


Figure 35

An interrupted thread tap must have an odd number of lands to generate this pattern, although taps with an even number of lands can be interrupted by removing teeth at random. Interrupted threads reduce friction, permit better lubrication and allow more space for the passage of chips. The chip load per tooth, however, is greater, which under some circumstances is beneficial and under others undesirable. Usually the chamfered portion of an interrupted thread tap is left with full threads. This reduces the possibility of reaming if no lead screw is used.

The added difficulties encountered when tapping taper pipe threads make tap design even more important than when straight threads are tapped. For this reason, a pipe tap for steel and aluminum, shown in Figure 36, has been specifically designed for these materials.



Figure 36

Similarly, there is available a taper pipe tap for cast iron and brass.

One of the most frequent causes of fluid or gas leakage is wavy threads in the internal fitting. A wavy thread, from visual inspection, appears to be multi-sided rather than round. Contributory factors may be cast or forged fittings with out of round holes or equipment with loose spindles, particularly if a sharp tap is used. As an expedient, if the cause is a loose spindle, a used tap which has been dulled by wear or deliberately dulled by stoning may eliminate the difficulty.

Dryseal Taper Pipe Taps

Taper pipe fittings with NPT threads may engage along the sidewalls of the threads without contact at crest and root. When this happens, a spiral clearance may extend from one end of the engagement to the other and thus allow leakage. When low pressures are involved, it has been the custom to assemble fittings with a lute or sealer which fills this clearance when it exists and, as a result, the joint becomes pressure tight to the extent necessary.

With certain refrigerants and fuels, a sealer cannot be used and to meet this requirement the American Standard Dryseal Taper Pipe Thread (NPTF) was developed. This thread requires very close tolerances at the crests and roots of both external and internal threads, with a truncation at the root of each so that fittings, when assembled hand tight, will contact at crests and roots before the flanks. Then, with the added wrench take-up, the flanks will meet and metal may actually be moved at the crest and root, depending on the extent of the interference. Thus, all possibility of spiral leakage is eliminated. Gaging requirements for the NPTF pipe threads are considerably more exacting than for NPT threads.

American National Dryseal Taper Pipe Taps (NPTF) will tap internal American National Dryseal Taper Pipe Threads.

Taper Reaming Before Tapping

The majority of internal pipe threads, although tapered, are generated by a taper pipe tap in a hole which is drilled or cored straight. This requires several threads to cut more than 100% thread. In Figure 37, the amount of metal removed by a taper pipe tap, in a straight hole, including the excess over 100% thread (a) is compared to the amount removed by a straight pipe tap tapping 100% thread (b).

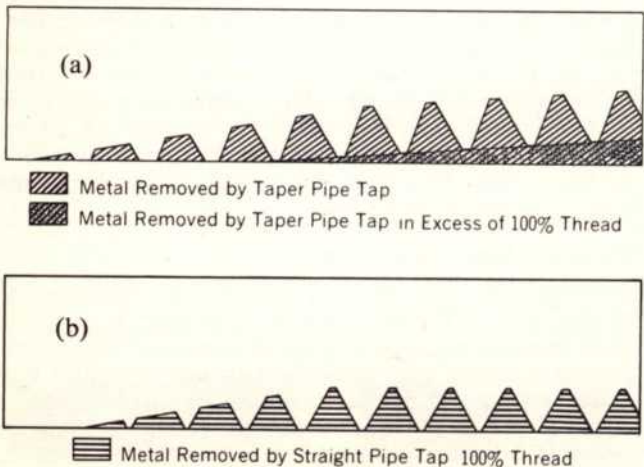


Figure 37

This condition may be relieved by taper reaming prior to tapping. Besides improving tapping conditions, resulting in longer tap life, this is helpful when producing threads to meet NPTF gaging specifications which require a close check of the taper at the minor diameter.

Sharpening

The information relative to the sharpening of straight thread taps is equally applicable to taper pipe taps, except that cutting face grinding is necessary. A few chipped threads do not mean that the tap is of no further use. Chipped threads, however, should be removed as soon as they appear. This does not impair the performance of the tap and in some cases it may even improve it by a partial simulation of the interrupted thread principle.

Tap drill sizes for pipe taps are shown below.

Hole Sizes For Pipe Taps

Nominal Pipe Size	TAPER*				Straight	
	Without Reamer		With Reamer			
	Drill	Decimal	Drill	Decimal	Drill	Decimal
1/16	D	.246	1 5/64	.234	1/4	.250
1/8	R	.339	2 1/64	.328	1 1/32	.344
1/4	7/16	.4375	2 7/64	.422	7/16	.438
3/8	3 7/64	.5781	9/16	.562	3 7/64	.578
1/2	4 5/64	.7031	1 1/16	.688	2 3/32	.719
3/4	5 9/64	.9219	5 7/64	.891	5 9/64	.922
1	1 5/32	1.156	1 1/8	1.125	1 5/32	1.156
1 1/4	1 1/2	1.500	1 1 5/32	1.469	1 1/2	1.500
1 1/2	1 4 7/64	1.734	1 4 5/64	1.703	1 3/4	1.750
2	2 7/32	2.219	2 3/16	2.188	2 7/32	2.219
2 1/2	2 5/8	2.625	2 1 9/32	2.594	2 2 1/32	2.656
3	3 1/4	3.250	3 7/32	3.219		
3 1/2	3 3/4	3.750				
4	4 1/4	4.250				

*These values may be used for NPT and NPTF taps.

Where the ASA B2.1 and ASA B2.2 (See Page 76) show only minor differences for NPT and NPTF drilled hole sizes, the smaller value is shown in the table above.

Methods of inspection vary. Care should be taken to use a tap drill which will make it possible to tap correct thread specifications. Sizes given for tapping without reaming give a full thread for only the first two or three threads.

TAPPING DIFFICULT-TO-MACHINE MATERIALS

Developments in jet engines, high speed aircraft, missiles, etc., have brought into being new materials and new forms of previously existing materials designed to withstand the extreme stress, heat and strength problems necessary for these applications. These new materials are classed as "difficult-to-machine" in comparison with the common types of steels, irons and light alloys. The problems of tapping them are subdivided into the following general classifications:

1. Titanium and Titanium Alloys
2. High Tensile Steels (Super-Strength Steels)
3. High Temperature Alloys (Superalloys)
4. Stainless Steels

Titanium and Titanium Alloys

Titanium and titanium alloys vary widely in tapping characteristics between the softer pure titanium and the harder alloys. The relatively recent development and use of titanium and its alloys, plus the accelerated programs in their application, have resulted in diversified opinions concerning the preferred procedures for tapping these materials.

The taps that have been successfully used include straight flute hand and spiral pointed designs. It is generally agreed that the friction should be minimized as much as possible. This can be accomplished by one or more of the following features: eccentric relief in the threads, additional chamfer relief, interrupted threads, additional back taper, narrow lands or axial grooves the full length of each land.

Eccentric pitch diameter relief with either full or interrupted threads has been found to be the most successful design feature for tapping titanium.



Where specifications require complete threads close to the bottom of the hole, a series of two or three taps with successively shorter chamfers may be required.

Wear and galling can be minimized by the application of a hard case surface treatment to the taps. An oxide treatment over the hard case may further reduce galling.

The tapping speeds used for the various alloys must be low, ranging from approximately 40 sfm for the softer grades down to 10 sfm or less for the hardest and toughest grades. The depth of the hole tapped will also influence cutting speed, with deeper holes requiring lower speeds.

The cutting fluid should be a light base oil or a mineral oil with lard. It should be applied in sufficient quantity to flood the tap throughout the tapping operation.

High Tensile Steels (Super-Strength Steels)

Tapping the high tensile steels in the hardness range 45 to 55 Rockwell C varies in difficulty with the analyses. AISI 4340 can be tapped in the 50 to 54 range, but a high chromium analysis is much more difficult to tap at the same hardness. This condition is probably associated with the presence of many hard carbides in a high strength matrix that causes increased resistance to a tool.

Tap recommendations for these materials must include careful consideration of chip load per tooth. The best results have been obtained with four flute taps having a 6-8 thread chamfer. Taps having fewer lands should not be used for materials in this hardness range. A flute form that provides a cutting face angle approximately radial (0° rake) should be used. The threads should not have eccentric relief, and the chamfer relief should be kept at a minimum in order to maintain as strong a cutting edge as possible.

Every possible means should be used to reduce the stress in cutting. The tapping speed should not exceed 10 sfm. The NF series of thread pitches should always be used with as low a percentage of thread as possible. The tapping depth should be held to a maximum of $1\frac{1}{2}$ tap

diameters. It is desirable to use a vertical machine with the tap held in a floating driver having a radial float only.

Bottoming holes are most difficult to tap and should be avoided if at all possible. The short chamfer required results in excessive stresses on the chamfer due to the greater chip load per tooth and causes rapid tap failure.

Taps made from the regular grades of high speed steel, with supplementary nitride and oxide treatments, will give good results in a majority of cases. Where there is extreme tapping difficulty, the special alloy high speed steels may show satisfactory gains in tool life.

Tapping fluids should be used in all cases, and should be activated sulpho-chlorinated oils, generously applied.

High Temperature Alloys (Superalloys)

A majority of high temperature alloys have work hardening characteristics, and both tap design and operating techniques must be guided by this factor. Their hardness and resistance to cutting varies with the particular group of alloys used, so that cutting speeds and, to some extent, tap design must be tailored to the particular alloy that is being tapped.

Taps having a cutting face hook of from 0° to 10° positive should be used. The harder the material, the lower the hook. The chamfer should be not more than 3 to 5 threads so as to produce chips that are thick enough to undercut any work hardened surface. The bearing surfaces on the tap lands should be as narrow as possible to minimize any work hardening effect due to rubbing.

Practical tapping speeds are low compared to conventional practice for mild steel, but must be varied with the hardness of the materials. In extreme cases, the practical tapping speeds are in the 5 to 10 sfm range.

Fine pitch threads and the lowest possible percentage of thread height are recommended. Length of thread in the tapped hole should also be kept as short as practicable, and should not exceed $1\frac{1}{2}$ times the tap diameter. Bottoming holes should be avoided.

Activated sulpho-chlorinated oils should be used generously as tapping fluids.

Stainless Steels

The tapping of the free machining grades of stainless steels is not much more difficult than the problems of tapping mild steels. However, those analyses that are generally difficult to machine are even more difficult to tap because the limitations of the tapping process emphasize the problems of metal removal and chip disposal.

The use of fine pitch threads and the largest permissible tap drill diameter are recommended for all stainless steels. Through holes are much easier to tap than blind holes because the chips can be forced ahead of the tap and because the tap can complete its cutting action before being reversed. Blind holes should be avoided wherever possible not only because of lack of chip room but also because of the problem of breaking the chips when the tap is reversed.

All taps for stainless steels should be made to cut as freely as possible consistent with the part specifications. Grooved, coneccentric or narrowed lands are sometimes used to minimize contact. Interrupted thread taps may be used where the increased chip thickness is not a problem.

Chordal hook angles from 12° to 15° are generally best on the softer grades of stainless but the harder grades are tapped best with taps of 10° to 12° hook. Spiral pointed taps produce good results in through and open holes up to $\frac{1}{2}$ " in diameter. In larger sizes, straight flute taps usually prove satisfactory.

The chamfer should not exceed 3 to 5 threads so that the chips are thick enough to get the cutting edges under any previously work hardened surface.

Tapping speeds in the order of 10 to 40 sfm are generally used, depending on the machinability of the material being tapped.

Frequent resharpening to maintain sharp cutting edges is vital particularly when tapping the work hardening grades.

Activated sulphurized and chlorinated oils are the recommended coolants and they should be applied copiously.

Most taps for stainless steel are made of regular high speed steel. Supplementary surface treatments have proven helpful in many cases because of the added abrasion resistance.

The specific recommendations for tapping the four classifications of "Difficult-to-Machine Materials" are shown below.

Tap Design and Tapping Suggestions for Tapping Difficult-to-Machine Materials

	TAP STYLE	RELIEF	HOOK	SURFACE TREATMENT	SPEED	LUBRI- CANT
High Temper- ature Alloys (Superalloys)	Straight Flute Plug or Spiral Point 3 - 5 Thread Maximum Chamfer	No Radial Thread Relief Regular Chamfer Relief	0 - 10°	Oxide or Nitride and Oxide	5 to 10 SFM	Activated Sulpho- Chlorinated Oil
Stainless Steels	Straight Flute Plug or Spiral Point 3 - 5 Thread Maximum Chamfer	Back Taper Con- centric Thread Relief Grooved or Narrow Lands	10 - 15°	Oxide or Nitride and Oxide	10 to 40 SFM	Activated Sulpho- Chlorinated Oil
Titanium and Titanium Alloys	Straight Flute Plug or Spiral Point Regular or Inter- rupted Thread	Eccentric Thread Relief Extra Chamfer Relief and Back Taper	3 - 8°	Nitride, Oxide or Both	40 SFM for soft grades to 10 SFM for hard	Activated Sulpho- Chlorinated Oil
High Tensile Steels (Super- Strength Steels)	4 Straight Flute 6 - 8 Thread Chamfer	No Radial Relief Minimum Back Taper	0°	Nitride and Oxide	10 SFM	Activated Sulpho- Chlorinated Oil

When short chamfers are required for tapping materials with an RC hardness of 40 or over, serial taps are indicated.



STRENGTH OF TAPPED HOLES

The strength of tapped holes becomes an important consideration to a tool engineer or tap user only when it creates tapping problems.

It is customary to design fastener applications so that the externally threaded member will break before either the external or internal thread strips and standard bolts and nuts are dimensioned so that failure will occur in this manner. Obviously this relationship can be maintained proportionately with relatively strong or weak assemblies.

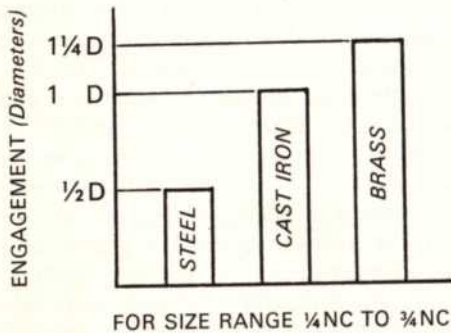
The majority of industrial applications of screw threads involve a standard bolt or screw and either a standard nut or an internally threaded product or component. In the latter case, the internal thread need be no stronger than a commercial nut of the same diameter and pitch. If a stronger external member is used, the strength of the tapped hole must be increased accordingly. This can be done by using stronger material, a greater percentage of thread or a longer thread. All of these conditions tend to increase tapping difficulties. Stronger material is tougher or harder and more difficult to cut, a greater percentage of thread produces a heavier chip and tapping longer threads develops a greater volume of chips.

The material selection is frequently beyond the control of the fabricator but where a tough or hard metal has been designated, the specifications of thread depth and length of engagement should be liberalized. The Chart (Figure 9) shown on Page 28 illustrates the relation between tapping torque and depth of thread. It is apparent that the increase in torque required to tap a thread with more than 60% depth is substantial. For example, an increase in thread depth from 60% to 72% in AISI 1020 steel requires twice the torque and it has been shown that the strength of the thread was increased by only 5%.

Because of material variations, it is difficult to compute lengths of engagement and depth of thread by formula but charts with empirical data which are available serve the purpose very well.

Strength of Tapped Holes

The chart (Figure 38) shows the engagement necessary to withstand a shearing stress of 150,000 P.S.I. in steel, cast iron and brass.



Approximate Engagement necessary with 75% Thread in Different Materials to withstand a Shearing Stress of 150,000 P.S.I.

Figure 38

Figure 39 illustrates the influence of thread depth on stripping strength for different materials and emphasizes that, with all other factors constant there is little difference in strength between threads with 75% and 100% depth.

The theory of the strength of screw threads and the data involved in the design of screw thread fasteners are complex subjects. The brief outline above is intended only to emphasize, to those responsible for both the design and production of screw threads, that adequate rather than over-engineering will effect immeasurable savings in tapping costs.

Stripping Load for Different Percentages of Thread (Threads $\frac{1}{4}$ - 28; length of engagement $\frac{3}{16}$ ")

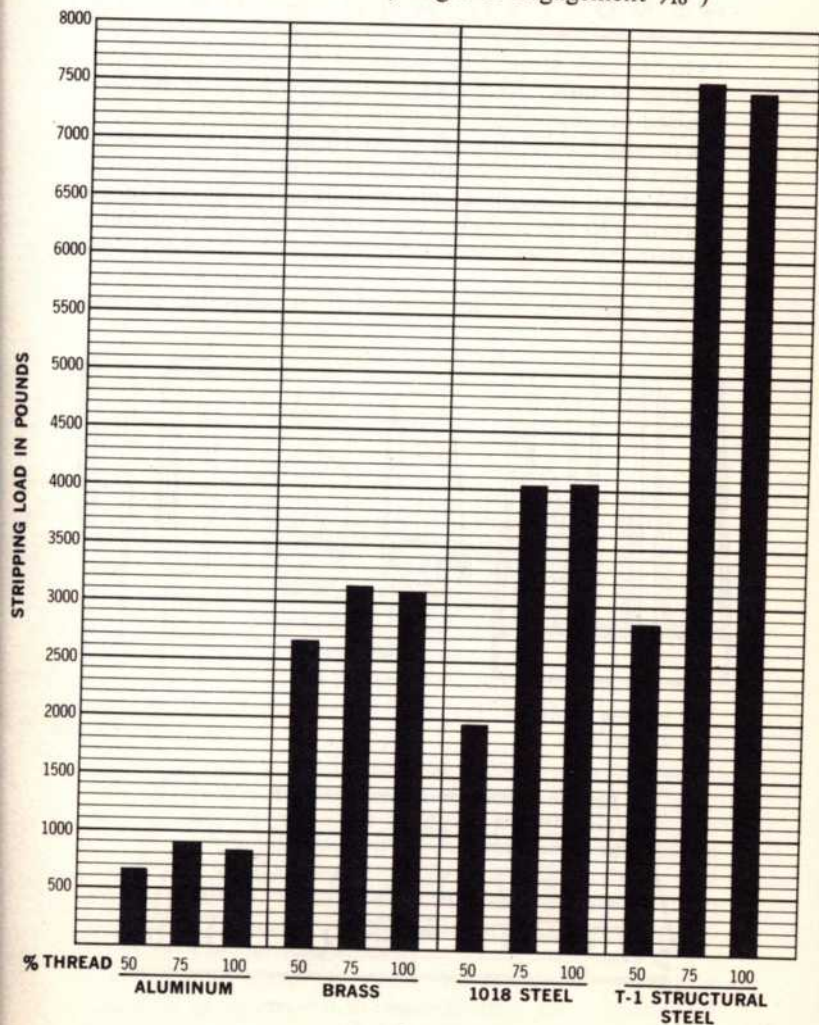
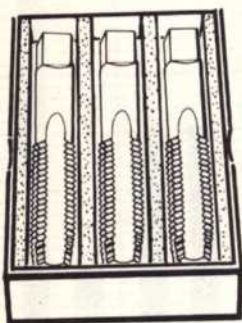


Figure 39

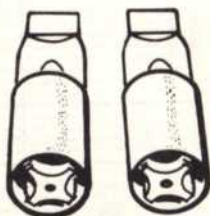
H A N D L I N G A N D S T O R A G E

Extreme care should be exercised in the handling and storage of taps; otherwise, cutting edges which become bruised or chipped may result in an early tap failure or in the production of threads of poor quality. One damaged thread is enough to cause trouble. Taps should be protected at all times, not only in storage but during transportation to and from the work or to and from a grinding department. They should be separated and kept dust free if possible.

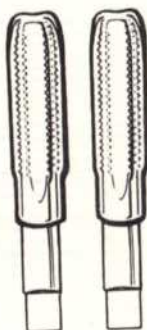
The following six illustrations show suggested ways to protect taps:



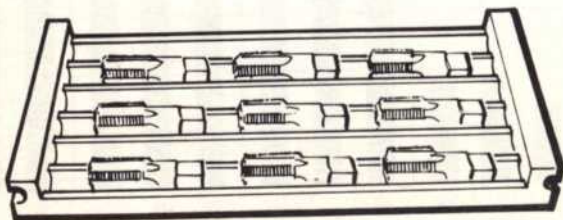
*The original package,
if available.*



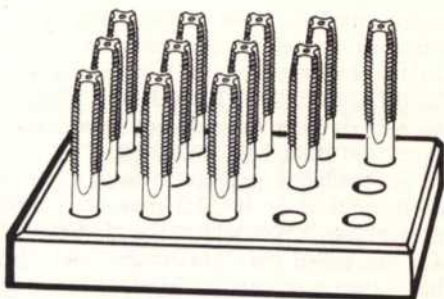
*Protective tubes
or sleeves of
cardboard or plastic.*



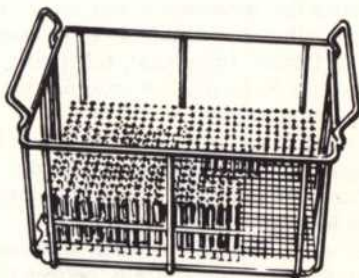
*Plastic or similar
protective coating.*



*Wooden trays or drawers
with dividers to isolate each tap.*



*Wooden blocks with holes
for inserting the tap shanks*



*Wooden or plastic coated wire racks
designed for the range of sizes used.*

Taps should never be packed loosely in a container without protection even when dull and awaiting resharp-ening. Chipping of teeth resulting from this practice requires flute grinding or, if it goes unnoticed, may cause immediate failure of the resharpened taps. Neat and orderly storage, clearly showing tap size and style, reduces the chance of issuing an incorrect tap for the job. This could be extremely costly.

The cost of good practice in handling and storage is negligible compared to the loss of time, tool failure and product spoilage which it may prevent.

MISCELLANEOUS TABLES

The "Basic" Point in Thread Measurement

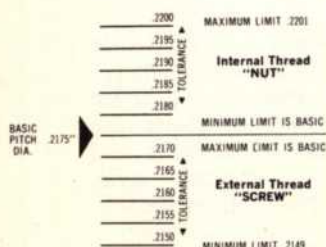
All measurements must have a controlling point or base from which to start. In the case of a screw thread, this control point is called the **BASIC** or theoretically correct size, which is calculated on the basis of a full form thread. Thus, on a given screw thread, we have the Basic Major Diameter, the Basic Pitch Diameter, and Basic Minor Diameter.

While it is impossible in practice to form screw threads to their precise theoretical or BASIC sizes, it is possible and practical to establish limits which the deviation must not exceed. These are called the "Maximum" and "Minimum" Limits. If the product is no smaller than the "Minimum Limit" and no larger than the "Maximum Limit," then it is within the size limits required. This difference between the Maximum and Minimum Limits is the **TOLERANCE**.

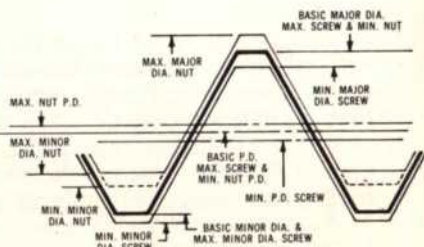
In actual practice the Basic size is not necessarily between the Maximum and Minimum Limits. In most cases, the Basic Size is one of the Limits. In general, tolerances for internal threads will be above Basic and for external threads, below Basic. See the example below.

For graphic representation, the Basic Pitch Diameter is commonly designated by a line with variations from it indicated by shorter lines spaced to represent a numerical scale, as shown on the left half of the sketch below.

On an actual screw thread, the Basic Dimensions would follow the contour of the theoretically perfect thread, as on the right.



1/4-20 NC Class 3 Thread



Thread Measurement Nomenclature

Constants for Finding Pitch Diameter and Minor Diameter of Screw Threads

To find the basic pitch diameter or basic minor diameter of any screw thread, subtract the constant for the number of threads per inch from the basic major diameter.

Threads per Inch	Constants for Finding Basic Pitch Diameter			Constants for Finding Basic Minor Diameter		
	National Thread	Whitworth Thread	Theoretical V	National Thread	Whitworth Thread	Theoretical V
80	.00812	.00800	.01083	.01624	.01601	.02165
72	.00902	.00889	.01203	.01804	.01786	.02406
64	.01015	.01000	.01353	.02030	.02001	.02706
60	.01083	.01067	.01443	.02165	.02134	.02887
56	.01160	.01144	.01546	.02320	.02286	.03093
50	.01299	.01281	.01732	.02598	.02562	.03464
48	.01353	.01334	.01804	.02706	.02668	.03608
44	.01476	.01455	.01968	.02952	.02910	.03936
40	.01624	.01601	.02165	.03248	.03202	.04330
36	.01804	.01779	.02406	.03608	.03558	.04811
32	.02030	.02001	.02706	.04059	.04002	.05413
30	.02165	.02134	.02887	.04330	.04268	.05773
28	.02320	.02287	.03093	.04639	.04574	.06186
27	.02406	.02372	.03208	.04812	.04742	.06416
26	.02498	.02463	.03331	.04996	.04926	.06662
24	.02706	.02668	.03608	.05413	.05336	.07217
22	.02952	.02911	.03936	.05905	.05821	.07873
20	.03248	.03202	.04330	.06495	.06403	.08660
18	.03608	.03557	.04811	.07217	.07114	.09623
16	.04059	.04002	.05413	.08119	.08004	.10825
14	.04639	.04574	.06186	.09279	.09147	.12372
13	.04996	.04926	.06662	.09993	.09851	.13323
12	.05413	.05336	.07217	.10825	.10672	.14434
11½	.05648	.05568	.07531	.11296	.11132	.15062
11	.05905	.05821	.07873	.11809	.11642	.15746
10	.06495	.06403	.08660	.12990	.12806	.17321
9	.07217	.07115	.09523	.14434	.14230	.19245
8	.08119	.08004	.10825	.16238	.16008	.21651
7	.09279	.09148	.12372	.18558	.18295	.24744
6	.10825	.10672	.14434	.21651	.21344	.28868
5½	.11809	.11642	.15746	.23619	.23284	.31492
5	.12990	.12807	.17321	.25981	.25613	.34641
4½	.14434	.14230	.19245	.28868	.28458	.38490
4	.16238	.16008	.21651	.32476	.32017	.43301
3½	.18558	.18295	.24744	.37115	.36590	.49487
3¼	.19985	.19702	.26647	.39970	.39404	.53294
3	.21651	.21344	.28868	.43301	.42689	.57733

FORMULA FOR OBTAINING TAP DRILL SIZES

$$\text{Outside Diam. of Thread} - \frac{.01299 \times \text{Amt. of Percentage of Full Thread}}{\text{No. of Threads per Inch}} = \text{Drilled Hole Size}$$

Note: Select nearest commercial stock drill.

PERCENTAGE OF FULL THREAD FOR OTHER DRILL SIZES

$$\text{No. of Threads per Inch} \times \left(\frac{\text{Outside Diam. of Thread} - \text{Selected Drill Diam.}}{.01299} \right) = \text{Amt. of Percentage of Full Thread}$$

American Standard and Dryseal Pipe Threads

BASIC DIMENSIONS—TAPER THREAD

Nominal Pipe Size Inches	Outside Diameter of Pipe— Inches	Threads per Inch	Pitch of Thread	Pitch Diameter at Gaging Notch— Basic— Inches	Thickness of Thin Ring Gage and Distance from Small End of Gaging Notch—Inches	Effective Thread External— Inches	Wrench Take Up— Inches	Total Length of Thread— Inches	Pitch Diameter at Small End of Effective Thread— Inches	Pitch Diameter at End of Pipe— Inches	Minor Diameter at End of Pipe— Inches	*Tap Drill Size	Minimum Hole Depth for Std. Pipe Tap
D	n	P	E ₁	L ₁	L ₂	L ₃	L ₄	E ₃	E ₀	K ₀			
1/16	27	.03704	.28118	.160	.2611	.1111	.3896	.26424	.27118	.2416		D	9/16
1/8	27	.03704	.37360	.1615	.2639	.1111	.3924	.35656	.36351	.3339		R	1 1/32
1/4	18	.05556	.49163	.2278	.4018	.1667	.5946	.46697	.47739	.4329		7/16	1 1/16
3/8	18	.05556	.62701	.240	.4078	.1667	.6006	.60160	.61201	.5676		3/8	1 1/8
1/2	14	.07143	.77843	.320	.5337	.2143	.7815	.74504	.75843	.7013		45/64	1 1/2
3/4	14	.07143	.98887	.339	.5457	.2143	.7935	.95429	.96768	.9105		5/8	1 3/4
1	11 1/2	.08696	1.23863	.400	.6828	.2609	.9845	1.19733	1.21363	1.1441		1 1/2	2 1/4
1 1/4	11 1/2	.08696	1.58338	.420	.7068	.2609	1.0085	1.54083	1.55713	1.4876		1 1/2	2 3/4
1 1/2	11 1/2	.08696	1.82234	.420	.7235	.2609	1.0252	1.77978	1.79609	1.7265		1 3/4	3 1/4
2	8	.12500	2.29627	.436	.7565	1.2500	1.0582	2.25272	2.26902	2.1995		2 1/2	4 1/2
2 1/2	8	.12500	2.76216	.682	1.1375	1.2500	1.5713	1.270391	2.71953	2.6195		3 1/2	5 1/2
3	8	.12500	3.38850	.766	1.2000	2.2500	1.6337	2.32500	3.34062	2.2406		4 1/2	6 1/2
3 1/2	8	.12500	3.88881	.821	1.2500	.250	1.6837	3.82188	3.83750	3.7375		5 1/2	7 1/2
4	8	.12500	4.38712	.844	1.3000	.250	1.7337	4.31875	4.33438	4.2344		6 3/4	8 1/2
5	8	.12500	5.44929	.937	1.4063	.250	1.8407	5.37511	5.39073	5.2907		7 3/4	9 1/2
6	8	.12500	6.50597	.958	1.5125	.250	1.9467	6.43047	6.44609	6.3461		8 3/4	10 1/2
8	8	.12500	8.50003	1.063	1.7125	.250	2.1467	8.41797	8.43359	8.3336		9 3/4	12 1/2
10	8	.12500	10.62094	1.210	1.9250	.250	2.3587	10.52969	10.54531	10.4453		10 3/4	14 1/2
12	8	.12500	12.61781	1.360	2.1250	.250	2.5587	12.51719	12.53281	12.3428		11 3/4	16 1/2

1) 2 1/2" NPTF and ANPT L₃=.375 E₃=2.69609 2) 3" NPTF and ANPT L₃=.375 E₃=3.31719

*Methods of inspection vary. Care should be taken to use a tap drill or taper reamer which can meet thread specifications. Sizes given permit direct tapping without reaming the hole, but only give a full thread for the first two or three threads. See columns K₀ and L₃.

Special Taps

Ground Thread—American National Form

GENERAL

The formulae shown below and the table on the opposite page are used in determining the limits and tolerances for ground thread taps having special diameter or special pitch or both, unless otherwise specified.

LEAD TOLERANCE

A maximum lead error of plus or minus .0005" in one inch of thread is permitted.

ANGLE TOLERANCE

Threads per inch	Error in half angle
4 to 5 1/2 incl.	20' Plus or Minus
6 to 9 incl.	25' Plus or Minus
10 to 80 incl.	30' Plus or Minus

FORMULAE

Max. Major Dia. = Basic plus A Max. Pitch Dia. = Min. plus D
 Min. Major Dia. = Max. minus B Min. Pitch Dia. = Basic plus C

In the above formulae:

A = Constant to add—

35% of the theoretical truncation for 4 to 5 threads per inch

40% for 5 1/2 to 12 threads per inch

45% for 13 to 80 threads per inch

To nearest .0005" for 8 or more threads per inch
 and to nearest .001" for less than 8 threads per inch

B = Major diameter tolerance

C = Amount over basic for minimum pitch diameter

D = Pitch diameter tolerance

NOTE: When the tap major diameter must be determined from a specified tap pitch diameter, the maximum major diameter equals the minimum specified pitch diameter *minus* constant C, *plus* the basic National single thread height, *plus* constant A.

Values for A, B, C and D

	A	B	C			D			
			To 5/8" incl.	Over 5/8" To 2 1/2" incl.	Over 2 1/2"	To 1" incl.	Over 1" To 1 1/2" incl.	Over 1 1/2" To 2 1/2" incl.	Over 2 1/2"
0	.0015	.0010	.0005	.0010	.0015	.0005	.0010	.0010	.0015
6	.0015	.0010	.0005	.0010	.0015	.0005	.0010	.0010	.0015
8	.0020	.0010	.0005	.0010	.0015	.0005	.0010	.0010	.0015
4	.0020	.0010	.0005	.0010	.0015	.0005	.0010	.0010	.0015
0	.0025	.0010	.0005	.0010	.0015	.0005	.0010	.0010	.0015
6	.0025	.0010	.0005	.0010	.0015	.0005	.0010	.0010	.0015
2	.0030	.0010	.0010	.0010	.0015	.0005	.0010	.0010	.0015
8	.0035	.0010	.0010	.0010	.0015	.0005	.0010	.0010	.0015
4	.0040	.0010	.0010	.0010	.0015	.0005	.0010	.0015	.0015
0	.0050	.0010	.0010	.0010	.0015	.0005	.0010	.0015	.0015
8	.0055	.0010	.0010	.0010	.0015	.0005	.0010	.0015	.0015
6	.0060	.0010	.0010	.0010	.0015	.0005	.0010	.0015	.0020
4	.0070	.0010	.0010	.0015	.0015	.0005	.0010	.0015	.0020
3	.0075	.0010	.0010	.0015	.0015	.0005	.0010	.0015	.0020
2	.0075	.0010	.0010	.0015	.0015	.0005	.0010	.0015	.0020
1	.0080	.0010	.0010	.0015	.0020	.0005	.0010	.0015	.0020
0	.0090	.0015	—	.0015	.0020	.0005	.0010	.0015	.0020
9	.0100	.0015	—	.0015	.0020	.0005	.0010	.0015	.0020
8	.0110	.0015	—	.0015	.0020	.0005	.0010	.0015	.0020
7	.0120	.0020	—	.0015	.0020	.0010	.0010	.0020	.0025
6	.0140	.0020	—	.0015	.0020	.0010	.0010	.0020	.0025
5 1/2	.0160	.0025	—	.0015	.0020	.0010	.0015	.0020	.0025
5	.0160	.0025	—	.0015	.0020	.0010	.0015	.0020	.0025
4 1/2	.0170	.0025	—	.0015	.0020	.0010	.0015	.0020	.0025
4	.0190	.0025	—	.0015	.0020	.0010	.0015	.0020	.0025

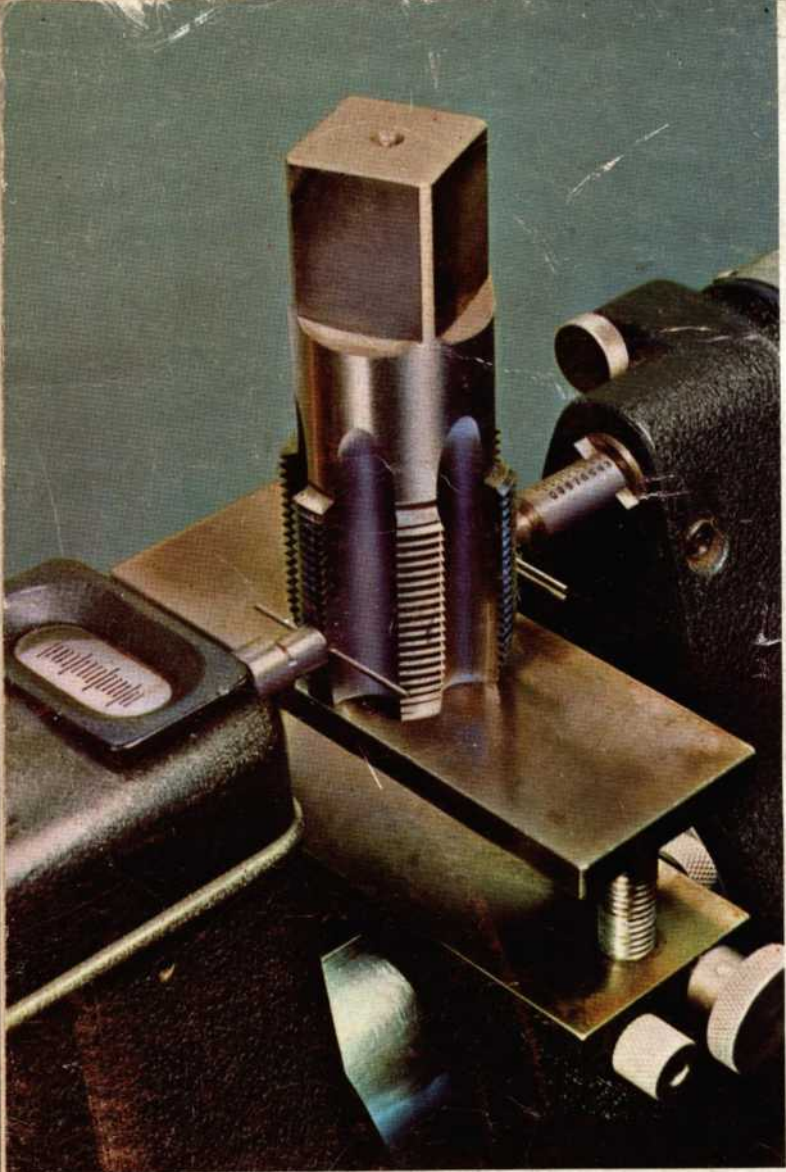
For intermediate pitches use value for next coarser pitch.

Additional Reference Material

For a more comprehensive study of screw threads and the use and care of taps, the following publications are available:

Metal Cutting Tool Handbook—Revised 1965 Drilled Holes for Tapping—Revised 1959 Standards and Dimensions for Taps and Dies —Revised 1964	<i>METAL CUTTING TOOL INSTITUTE</i>																						
Screw-Thread Standards for Federal Services, Handbook H-28, Parts I, II and III, 1957 Supplement to Screw-Thread Standards for Federal Services, 1963	<i>U.S. DEPARTMENT OF COMMERCE</i>																						
<table border="0"> <tr> <td>Unified Screw Threads</td><td>B1.1 -1960</td></tr> <tr> <td>Acme Screw Threads</td><td>B1.5 -1952</td></tr> <tr> <td>Nomenclature, Definitions and Letter Symbols for Screw Threads }</td><td>B1.7 -1965</td></tr> <tr> <td>Stub Acme Screw Threads</td><td>B1.8 -1952</td></tr> <tr> <td>Buttress Screw Threads</td><td>B1.9 -1953</td></tr> <tr> <td>Unified Miniature Screw Threads</td><td>B1.10-1958</td></tr> <tr> <td>Microscopic Objective Thread</td><td>B1.11-1958</td></tr> <tr> <td>Class 5 Interference-Fit Thread</td><td>B1.12-1963</td></tr> <tr> <td>Pipe Threads (Except Dryseal)</td><td>B2.1 -1960</td></tr> <tr> <td>Dryseal Pipe Threads</td><td>B2.2 -1960</td></tr> <tr> <td>Taps—Cut and Ground Threads</td><td>B94-Revised 1965</td></tr> </table>	Unified Screw Threads	B1.1 -1960	Acme Screw Threads	B1.5 -1952	Nomenclature, Definitions and Letter Symbols for Screw Threads }	B1.7 -1965	Stub Acme Screw Threads	B1.8 -1952	Buttress Screw Threads	B1.9 -1953	Unified Miniature Screw Threads	B1.10-1958	Microscopic Objective Thread	B1.11-1958	Class 5 Interference-Fit Thread	B1.12-1963	Pipe Threads (Except Dryseal)	B2.1 -1960	Dryseal Pipe Threads	B2.2 -1960	Taps—Cut and Ground Threads	B94-Revised 1965	<i>AMERICAN STANDARDS ASSOCIATION</i>
Unified Screw Threads	B1.1 -1960																						
Acme Screw Threads	B1.5 -1952																						
Nomenclature, Definitions and Letter Symbols for Screw Threads }	B1.7 -1965																						
Stub Acme Screw Threads	B1.8 -1952																						
Buttress Screw Threads	B1.9 -1953																						
Unified Miniature Screw Threads	B1.10-1958																						
Microscopic Objective Thread	B1.11-1958																						
Class 5 Interference-Fit Thread	B1.12-1963																						
Pipe Threads (Except Dryseal)	B2.1 -1960																						
Dryseal Pipe Threads	B2.2 -1960																						
Taps—Cut and Ground Threads	B94-Revised 1965																						





BAY STATE TAP AND DIE
Division of **THE CLEVELAND TWIST DRILL CO.**
P.O. Box 6656 • Cleveland, Ohio 44101